INTRODUCTION

Viscosity is an important fluid property which permits the development of shear stresses and resistance to flow. For static fluids, only normal stress or pressure exists.

The ideal fluid was defined to be inviscid i.e. the viscosity has no effect. Thus, there were no frictional forces between moving fluid layers or between the fluid and bounding walls.

The flow of a real fluid is more complex than that of an ideal fluid, owing to the phenomena caused by the existence of viscosity.

Viscosity introduces resistance to motion by causing shear and friction forces between fluid particles and boundary walls.

For flow to take place, work must be done against resistance forces, and in the process, energy is converted to heat. The inclusion of viscosity also allows the possibility of two physically different flow regimes. The effects of viscosity on the velocity profile also render invalid the assumption of uniform velocity distribution. Although the Euler equations may be altered to include shear stresses of a real fluid, the result is a set of partial differential equations to which no general solution is known.

Laminar and Turbulent Flow

In laminar flow, agitation of fluid particles is of a molecular nature only, i.e. at a length scale of the order the mean free path of the molecules. On the usual macroscopic scale of observation, fluid particles appear to be constrained to motion in essentially parallel paths by the action of viscosity, Fig. 1.la.

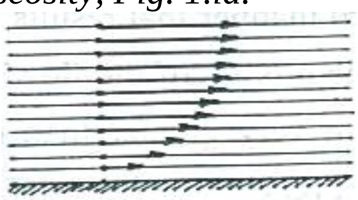


Fig.1.1a

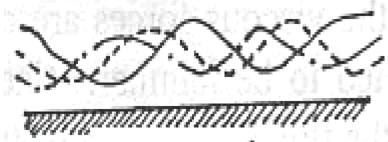


Fig. 1.1b

The shearing stress between adjacent layers in the laminar flow simple parallel flow) is completely defined by $\tau = \mu \frac{du}{dy}$ *, see Fig.1.2*

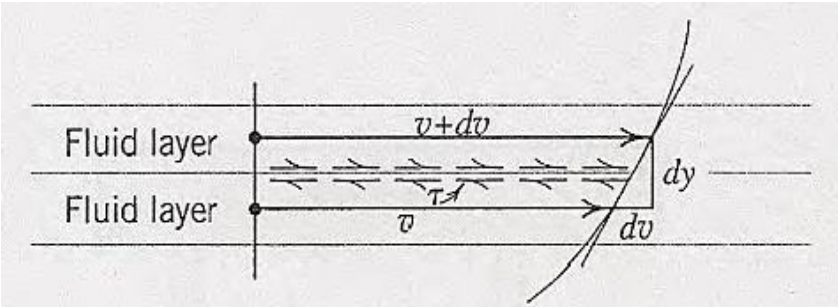


Fig. 1.2

If the laminar flow is disturbed by wall roughness or some other obstacle, the disturbances are rapidly damped by viscous action, and downstream the flow is smooth again. Therefore a laminar flow is stable against such disturbances, but a turbulent flow is not.

In turbulent flow, fluid particles don't remain in layers, but move in a heterogeneous fashion through the flow, sliding past other particles and colliding with some in an entirely haphazard manner that results in a rapid and continuous macroscopic mixing of the flowing fluid, with length scales which are very greater than the molecular, scales in laminar flow, Fig. 1.1 b

The random motion and the observed eddies in a turbulent flow suggest that both the inertia forces, associated with the accelerations during motion, and the viscous forces induced by the action of the viscosity, may be important. When the viscous forces are dominant, the flow might be expected to be laminar. When the inertia forces are dominant, the flow might well be turbulent.

Both laminar and turbulent flows were demonstrated by Reynolds, (1883), with an apparatus similar to that of Fig. 1.4.

REYNOLD'S EXPERIMENTS

The effects of viscosity cause the flow of a real fluid to occur under two different conditions or regimes: that of laminar flow and that of turbulent flow. Reynolds was the first one demonstrated the characteristics of these regimes, with an apparatus similar to that of Fig.1.1a

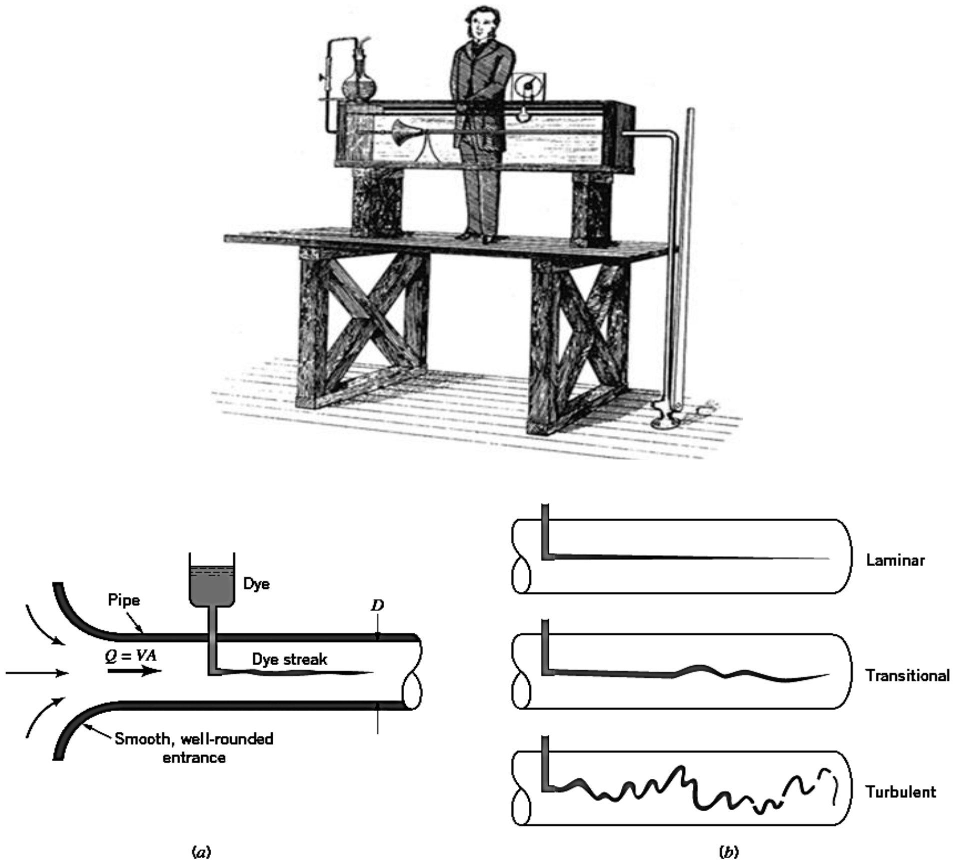


Fig. 1.4 (a) Reynolds Apparatus (b) different flow types

Water flows from a tank through a bell-mouthed glass pipe, the flow being controlled by the valve. A thin tube leading from a reservoir of dye has its opening within the entrance of the glass pipe.

Reynolds discovered that, for low velocities of flow in the glass pipe, a thin filament of dye issuing from the tube did not diffuse but was maintained intact throughout the pipe, forming a thin

straight line parallel to the axis of the pipe (Fig.1.4b). As the valve was being opened, however, and greater velocities were attained, the dye filament wavered and broke, eventually diffusing through the flowing water in glass pipe.

Since mixing of fluid particles during flow would cause diffusion of the dye filament, Reynolds deduced from his experiments that at low velocities this mixing was absent and that the fluid particles moved in parallel layers, or lamina, sliding past adjacent lamina but not mixing with them, this is the regime of laminar flow. Since at higher velocities the dye filament diffused through the pipe, it was apparent that mixing of fluid particles was occurring, or, in other words, the flow was turbulent. Laminar flow broke down into the turbulent flow at some critical velocity above that at which turbulent flow was restored to the laminar condition.

Reynolds was able to generalize his conclusions from his dye stream experiments by the introduction of a dimensionless term Re, later called the ***Reynolds Number****, which was defined by:*

$$\mathbf{Re} = \frac{\boldsymbol{Vd\rho}}{\boldsymbol{\mu}} = \frac{\boldsymbol{Vd}}{\boldsymbol{\nu}}$$

In which V is the mean velocity of the fluid in pipe, d is the diameter of the pipe, and $\boldsymbol{\rho}$, $\boldsymbol{\mu}$ *and* $\boldsymbol{\nu}$ *are the specific mass, dynamic viscosity and Kinematic viscosity of the fluid flowing therein.*

The upper limit of laminar flow is defined by $\boldsymbol{2100 < Re_{cr} < 4000}$*. The lower limit of turbulent flow, defined by the lower critical Reynolds number, is of greater engineering importance; it defines a condition below which all turbulence entering the flow from any source will eventually be damped out by viscosity. This lower critical Reynolds number thus sets a limit below which laminar flow will always occur; many experiments have indicated the lower critical Reynolds number to have a value of approximately 2100. Between Reynolds number 2100 and 4000 a region of uncertainty exits.*

The concept of a critical Reynolds number to the flow of any fluid in cylindrical pipes، one may predict that the flow will be

laminar if Re<2100 and turbulent if Re>4000. *However، critical Reynolds number is very much a function of boundary geometry.*

As conclusion, laminar flows (small Re) are characterized by low velocities, small length scales (e.g. small diameter pipes) and fluids with high kinematic viscosity. Turbulent flows (large Re) are characterized by high velocities, large length scales, and fluids of low kinematic viscosity.

1. Introduction to Laminar Flow

The various ways of classifying flows have been met in earlier notes. Below is a summary of the broad classifications:

Flow Description Steady/Unsteady the manner in which the fluid velocity varies with time.

In steady flow *the velocity is unchanging at any site with time. If you measure the fluid velocity at time* ***T=0*** *and you measure the velocity some time* $T=t_1$ *after, at the same place, then the velocity would be the same.*

In unsteady flow*, the velocities would not be the same.*

Uniform/Non-uniform *the manner in which the fluid velocity varies in space.*

In uniform flow*, the velocity is unchanging at any place with time. If you measure the velocity at a point, and then you measure the velocity at another point in the fluid, the velocities would be the same for any time at which the experiment is performed.*

In non-uniform flow*, the velocities at the two points would be different.*

Laminar/Turbulent, the manners in which fluid particles move relative to each other.

In laminar flow, individual particles of fluid follow paths that do not cross those of neighboring particles streamlines. Streamlines help in our understanding fluid flow.

Recall from previous notes that a streamline is an imaginary curve drawn through a mass of fluid. The streamlines at time t form the family of curves everywhere tangent to the velocity field at that time (every point on the streamline is tangent to the net

velocity vector). When the flow is steady, the fluid particles move along the streamlines. There can be no net flow across a streamline.

Flow Regimes

At low flow rates, fluids move in "layers". The velocity components of the individual fluid elements do not cross the streamlines. This type of behavior is called laminar flow (or streamlines flow). In pressure driven flows laminar flow is predominant only at low flow rates. At higher flow rates, the streamlines are disrupted by eddies moving in all directions. These eddies make the flow turbulent, and although they cross and re-cross the streamlines, the net velocity is such that, the net flow from eddies is zero.

Eddies form from contact of the fluid with a solid boundary or from two fluid layers moving at different speeds. As these eddies grow, the laminar flow becomes unstable and velocities and pressures in the flowing fluid no longer have constant or smoothly varying values.

In pipes, relatively large rotational eddy form in regions of high shear near the pipe wall. These degenerate into smaller eddies as energy is dissipated by action of viscosity. The presence of eddies means that the local velocity is not the same as the bulk velocity and that there are components of velocity in all directions.

The transition between laminar and turbulent flow is fuzzy. The behavior depends on entrance conditions and distance from the inlet. Often it is useful to speak of a transition region for flows that are neither laminar nor fully turbulent.

Distinguishing Features of Laminar Flow

−as the name suggests, the flow is characterized by layers of fluid traveling at different velocities.

− Individual particles are held in place by molecular forces that prevent them from wandering outside their streamline.

− Viscous forces predominate over inertia forces.

− Velocity gradients set up across the flow.

Occurrences

Associated with slow moving of viscous fluids; it is relatively rare in nature, but one example is flow of water through an aquifer. The velocities may be as low as a few meters per year.

The text treats several examples that have specific interest to engineering. We will only be looking at a few of these. We look at flows in circular pipes since they are the most commonly used shape.

Viscous Forces: Only Newtonian fluids are being considered in this course. Recall from the first few lectures that for such fluids, the shear stress developed between layers moving relative to each other was given by the following relation: $\tau = \mu \frac{du}{dy}$

*Where, **τ** is the shear stress parallel to the fluid motion, **du/dy** is the velocity gradient perpendicular in the transverse direction and **μ** is the dynamic viscosity. This relation is fundamental to our development of the relation.*

***Pressure forces**: Also of importance are the forces set up by pressure at the ends of the flow. Note that for steady, fully developed flow, there is no acceleration and so the total force is zero.*

Water at 15°C has a kinematic viscosity of 1.139×10^{-6} m^2/s and flows in a cylindrical pipe of 30 mm diameter. Calculate the largest flowrate for which laminar flow can be expected. What is the equivalent flowrate for air?

Solution:

Assume that the flow is one-dimensional and the flowrate

$$Q = AV \tag{4.4}$$

while the Reynolds number

$$\mathbf{R} = \frac{Vd}{\nu} \tag{7.1}$$

For this calculation, the kinematic viscosity of water and the pipe diameter are given as $\nu = 1.139 \times 10^{-6}$ m²/s and $d = 0.03$ m. Taking $\mathbf{R}_c = 2\,100$ as the conservative upper limit for laminar flow,

$$2\,100 = \frac{V(0.03)}{0.000\,001\,139}; \qquad V = 0.080 \text{ m/s} \tag{7.1}$$

$$Q_{\text{water}} = (0.080) \times \left(\frac{\pi}{4}\right)(0.03)^2 = 5.64 \times 10^{-5} \text{ m}^3/\text{s} \bullet \tag{4.4}$$

The water has the given viscosity at about 15°C so from Appendix 2 we have for air at zero altitude, which is at that temperature,

$$\nu_{\text{air}} = (\mu/\rho)_{\text{air}} = (1.789 \times 10^{-5} \text{ Pa·s})/(1.225 \text{ kg/m}^3) = 1.46 \times 10^{-5} \text{ m}^2/\text{s}$$

and

$$2\,100 = \frac{V(0.03)}{0.000\,014\,6}; \qquad V = 1.022 \text{ m/s} \tag{7.1}$$

$$Q_{\text{air}} = (1.022) \times \left(\frac{\pi}{4}\right)(0.03)^2 = 7.22 \times 10^{-4} \text{ m}^3/\text{s} = 13Q_{\text{water}} \bullet \tag{4.4}$$

2. Turbulent Flow and Eddy Viscosity:

The characteristics of turbulent flow can be listed as follows:

- Irregularity or randomness in time and space.
-Diffusivity or rapid mixing.
-High Reynolds number.
-Three-dimensional vortices fluctuations.
-Dissipation of the kinetic energy of the turbulence by viscous shear stresses.
- Turbulence is a feature of fluid flow.

The vorticity fluctuations symbolize the three-dimensional nature of the turbulence. The characteristic of dissipation makes it clear that there must be a continuous energy supply, from the mean flow to the turbulence, or it will decay.

To analyze turbulence it is useful to focus on fluid particles.

These particles are observed to travel in randomly moving fluid masses of varying sizes called "eddies" these cause at any point in the flow, a rapid and irregular pulsation of velocity about a well defined mean value. This may be visualized as in Fig. 1.5, where V is the time mean velocity which can be measured directly with a small pitot tube and $\boldsymbol{V'}$ *is the instantaneous velocity, such that:*$\boldsymbol{V = \dddot{V}'}$ *, the over bar denotes a time average value.*

The instantaneous velocity $\boldsymbol{V'}$ *may be considered to be composed of vector sum of the mean velocity and the components of pulsations* V_x *and* V_y*, which can be measured by -hot-wire anemometer and its record is shown in Fig. 1.5, which 'discloses no regular period or amplitude because of the random nature of turbulence. The shown records allow the definition of certain turbulence characteristics:*

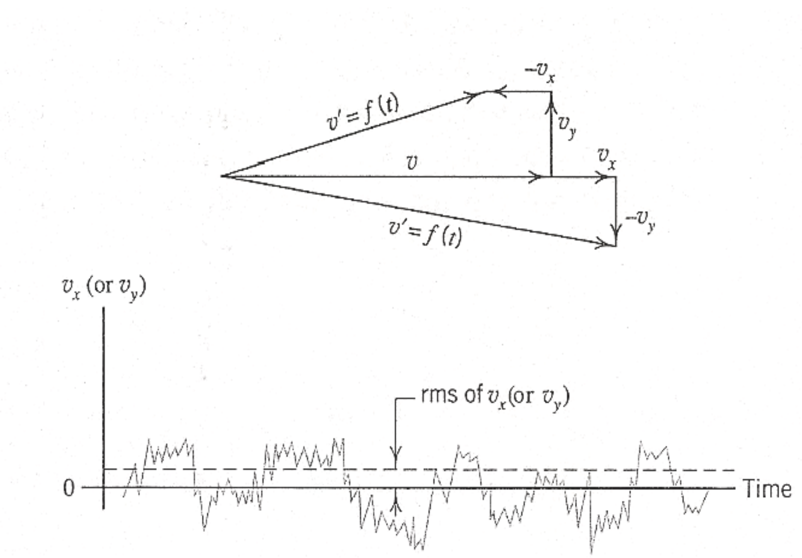


Fig. 1.5

-The root-mean square (r m s) values $\left(\dddot{V}_x^2\right)^{\frac{1}{2}}$ *or* $\left(\dddot{V}_y^2\right)^{\frac{1}{2}}$ *are a measure of the violence of turbulent fluctuations, i.e., the magnitude of departure of V' from V.*

-The relative intensity of turbulence $\left(\dddot{V}_x^2\right)^{\frac{1}{2}}/V$

-The scale of the turbulence, the mean time interval between

reversals in the sign of V_x (or V_y), because it is a measure of the size of the turbulent eddies passing the point.

Shearing stress in a simple parallel turbulent flow may be visualized by considering two adjacent points in a flow cross section, Fig. 1.6, at one of these points the mean velocity is V, at the other V + AV, where V & V + ΔV is taken to be the mean velocities of fluid layers. The turbulence velocity V_y represents the observed transverse motion of small fluid masses between layers in both directions. The fluid masses move towards layer with small mean velocity is slowed down, while the fluid masses move towards layer with higher mean velocity is speeded up, i.e. fluid masses momentum is changed during their transfer process (tending to speed up the slower layer and slow down the faster one), just as if there were a shearing stress between layers. Thus the existence of shearing stress in turbulent flow is deducible from momentum considerations.

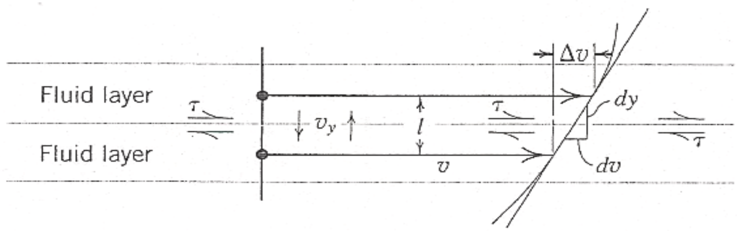


Fig. 1.6

*The turbulent shear stress was given by **Bossinesq (1877)** who followed the pattern of the laminar flow equation and wrote:*

$$\tau = \varepsilon \frac{dv}{dy}$$

*Where **ε** is the eddy viscosity, it is property of the flow (not fluid alone) which depends primarily on the structure of the turbulence.*
Prandtl (1926) derived the following relation for the eddy viscosity in the form:

$$\varepsilon = \rho l^2 (dv/dy)$$

*Where he termed the distance **l** the mixing length, which is the mean distance which small aggregations of fluid particles are*

transported by turbulence from region of one velocity to region of another.

*As near the boundary wall, the turbulence is strongly influenced by the wall (V_y, V_x = 0, fluctuation velocities of fluid particles due to turbulence, normal and along the direction of general mean motion), this let the mixing length **l** vary directly with distance from the wall y, where*

$$l = \kappa y$$

*Where K is von Kàrman constant, which has been determined by the use of experimental data, its nominal value is **K= 0.4**, then the turbulent shear stress yields:*

$$\tau = \rho \kappa^2 y^2 \left(\frac{dv}{dy}\right)^2$$

Show that, if the velocity profile in laminar flow is parabolic, the shear stress profile must be a straight line.

For a parabolic relation between v and y, $v = c_1 y^2 + c_2$. From Eq

$$\tau = \mu \frac{dv}{dy}$$

Therefore, because $dv/dy = 2c_1 y$, $\tau = \mu(2c_1 y)$ and clearly, $\tau \propto y$

PROBLEM 7.3

A turbulent flow of water occurs in a pipe of 2 m diameter. The velocity profile is measured experimentally and found to be closely approximated by the equation $v = 10 + 0.8 \ln y$, in which v is in metres per second and y (the distance from the pipe wall) is in metres. The shearing stress in the fluid at a point $\frac{1}{3}$ m from the wall is calculated from measurements of pressure drop (see Section 7.10) to be 103 Pa. Calculate the eddy viscosity, mixing length, and turbulence constant at this point.

The velocity gradient is needed and from the given profile

$$v = 10 + 0.8 \ln y, \qquad \frac{dv}{dy} = \frac{0.8}{y} = \frac{0.8}{(0.33)} = 2.4\ \text{s}^{-1}$$

at a distance of 0.33 m from the wall.

From Eq. 7.2, $\tau = \varepsilon \dfrac{dv}{dy}$ yields 103 Pa = 103 N/m² = $\varepsilon(2.4\ \text{s}^{-1})$,

$$\varepsilon = 42.9\ \text{Pa·s (Note } \mu_{\text{water}} \approx 10^{-3}\ \text{Pa·s)}$$

From Eq. 7.3, $\tau = -\rho\overline{v_x v_y} = \rho l^2 \left(\frac{dv}{dy}\right)^2$ yields (if $\rho = 1\,000 \text{ kg/m}^3$)

$$103 \text{ N/m}^2 = 1\,000 l^2 (2.4)^2, \qquad l = 0.134 \text{ m} \bullet$$

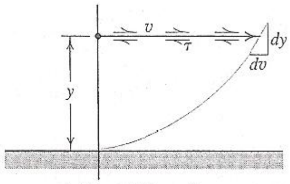
From Eq. 7.6, $\tau = \rho\kappa^2 y^2 \left(\frac{dv}{dy}\right)^2$ yields

$$103 = 1\,000\kappa^2 \left(\frac{1}{3}\right)^2 (2.4)^2, \qquad \kappa = 0.401 \bullet$$

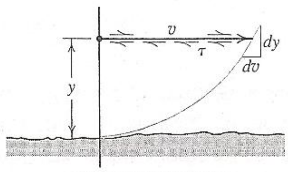
The magnitude of the eddy viscosity ε when compared with the viscosity μ (approximately 0.001 Pa·s) is of special interest in that it provides a direct comparison between the (large) turbulent shear and (small) laminar shear for the same velocity gradient. The mixing length, l, when compared with the pipe radius is found to be about 10% of the latter dimension; this is a nominal value of correct order of magnitude, as is the turbulence constant, κ.

3. Fluid Flow past Solid Boundaries.

For real fluid, experimental evidence shows that the velocity of the layer adjacent to the surface is zero (relative to the surface). This means that a velocity profile must show a zero value at the boundary. For laminar flow, surface roughness has no effect on the flow picture, Fig. ***8.6,***



(*a*) Smooth boundary



(*b*) Rough boundary

Fig. 7.6 Laminar flow over boundaries that *act* smooth.

Fig. 1.7

In turbulent flow, the presence of the boundary causes reduction of

mixing Length to zero value (i.e. the turbulence is completely extinguished) and a film of viscous flow (viscous sublayer) over the boundary results. A boundary surface is said to be smooth if its projection of protuberances are so completely submerged in the viscous sublayer and then they have no effect on the structure of turbulence, Fig.1.8.However, experiments have shown that roughness heights larger than about one-third of sublayer thickness will augment the turbulence and have some effect on the flow.

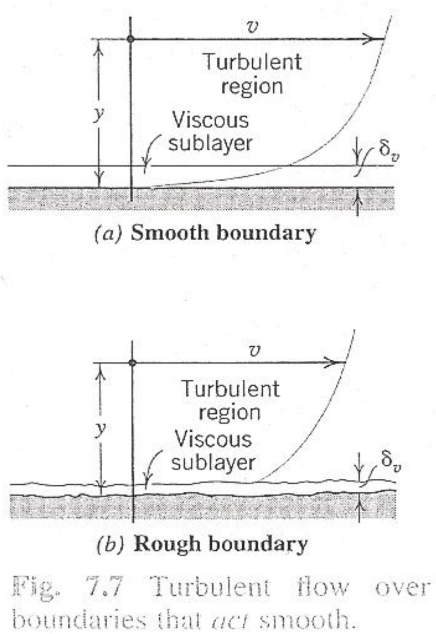


Fig. 1.8

EXTERNAL FLOWS

In external flow cases, energy or work is used typically to move the object through the fluid. In external flows, one is interested to determine the flow pattern (including the pattern of viscous effect) around an object (wing or flat plate, etc) immersed in the fluid, from which to determine the lift and drag (resistance to motion) on the object.

For internal flow, the focus is often not on lift and drag, but on energy or head losses, pressure drops and cavitation where energy is dissipated, because in internal flow, energy or work is

used to move fluid through passages.

4. Characteristics of the Boundary Layer.

Analysis made on the basis of ideal (inviscid) However streamlined bodies produces tow results:

- The calculated drag on the body is negligible, while the observed drag is not.

- The ideal fluid slips smoothly by the body, real fluid does not.

For fluid with small viscosity (e.g. air and water), Prandtl suggested the boundary layer concept to explain the resistance of the streamlined bodies Fig. 8.8, flat plates parallel to the flow ,airfoil and so forth. The essential point is that the frictional aspects of the flow are confined to the boundary layer and perhaps a wake behind the body, where the, flow is rotational, but outside the boundary layer, (the viscosity of the fluid is essentially inoperative), the flow is effectively frictionless and irrotational.

Boundary layer phenomena are visualized on a smooth flat plate parallel to the oncoming flow, Fig 1.9. On this plate the boundary layer may be either laminar or turbulent. With smooth plate leading edge, a laminar boundary layer is to be expected adjacent to the upstream portion of the plate since the boundary layer is thin, viscous action intense, and turbulence is inhibited. As explained in section 8.1, the Reynolds number is used to define the character of boundary layer, and we can define two Reynolds number depend on the used length scale, Re_x *(.on the base of distance from the plate leading edge x), and* $\boldsymbol{Re_\delta}$ *(on the base of boundary layer thickness* $\boldsymbol{\delta}$*), in the form :*

$$\mathbf{R}_x = \frac{V_o x}{\nu} \qquad \text{and} \qquad \mathbf{R}_\delta = \frac{V_o \delta}{\nu}$$

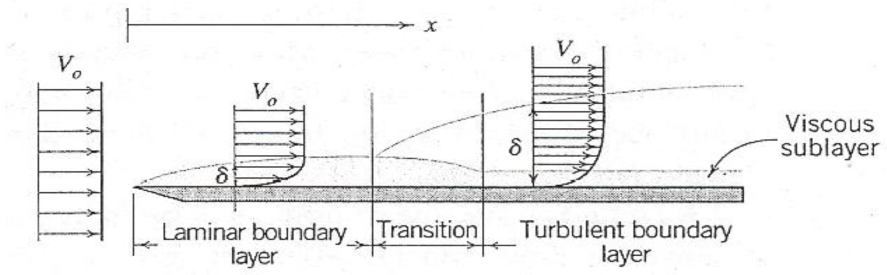


Fig. 7.8 Boundary layers on a flat plate.

Fig. 1.9

Experiments show that the typical critical values for ***Re_x=500 000*** *and* ***Re_δ = 3 900****, such that below these values, laminar boundary layer are to be expected. Whereas Reynolds numbers above these values the flow is transitioning to turbulent boundary layer with a thin viscous sub-layer. For excessive turbulence in the oncoming flow or a rough leading edge of the plate, it may imply a turbulent boundary layer beginning at the leading edge of the plate.*

The critical Reynolds number ***Re_δ*** *is also very sensitive to pressure gradient as follow:*

-A favorable gradient, pressure decreasing in the direction of flow (flow in nozzle), stabilizes the laminar boundary layer and ***Re_δ > 3900****.*

-Conversely, an unfavorable pressure gradient, pressure increasing in the direction of the flow, as in diffuser, causes early breakdown of the laminar flow and ***Re_δ < 3900****.*

For real fluid flow an engineering approach for solution of such flow problem is to solve first the outer problem of ideal fluid motion about the body, ignoring viscous effect entirely. Then using the "outer" solution values of velocity and pressure at the surface of the body as approximate values for the edge of the boundary layer, the "inner" viscous flow problem, is solved. For streamlined shapes, this procedure gives, from the "outer" solution, the pressure distribution (including an accurate estimate of the lift force, if any) and, from the "inner" solution, an estimate of the friction force or drag on the shape is calculated.

Now comparison between the streamlined body and flat plate boundary layers, Figs. 8.8 & 8.9 gives:

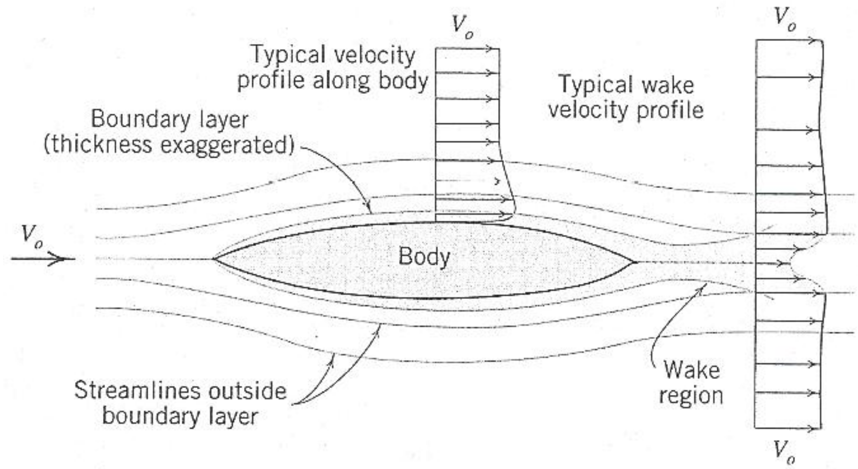


Fig. 7.9

Fig.1.10

- *Superficial similarity is noted immediately.*
- *For the streamlined body:*

** The surface curvature may affect the boundary layer development due to either internal effects or induced separation.*

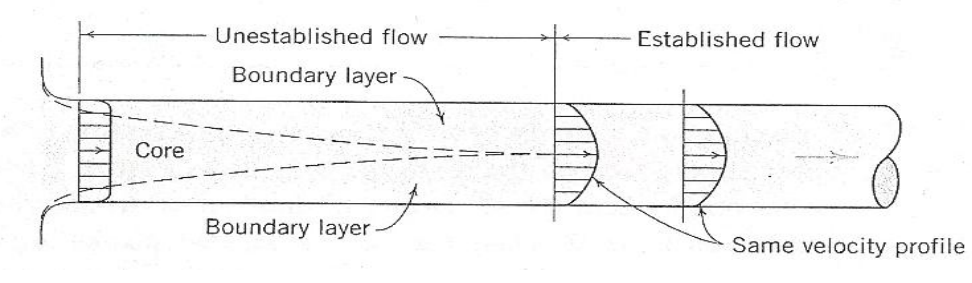
** The velocity just outside the boundary layer (in the irrotational flow) change continuously along the body because of the disturbance of the overall flow due to body finite width.*

Flow Establishment- Boundary Layers

It is considered here the simplest case, flow from a large reservoir into a cylindrical passage with a round entrance Fig.

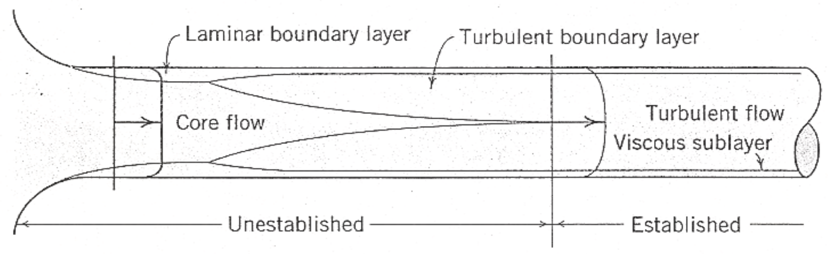
3.23. The boundary layer will start at the entrance and will grow, with the distance along the tube (accompanied by a diminishing core of irrotational fluid at the center of the passage. Thus the boundary layer steadily thickness until they meet and so envelop the whole flow, section-B, beyond which all velocity profiles will be identical. The flow is, .therefore established (developed) and is everywhere rotational. Downstream of section B, the turbulence of wall friction is felt throughout the flow field.

Of course, the flow in a boundary layer may be either laminar or turbulent. If the Reynolds number, $\mathrm{Re} = \frac{vd}{\nu}$ *, for the established flow is* ***less than 2100,*** *it may be said that the established laminar* ***flow*** *has resulted from the growth of laminar boundary layers. In this case the zone of establishment has a length given by:* $\frac{x}{d} = \frac{Re}{20}$*. For slightly higher* ***Re,*** *the flow may be laminar up to and past* $\frac{x}{d} = \frac{\mathrm{Re}}{20}$*, with subsequent transition to turbulent flow before the flow is truly established.*



In Re » 2100 *the boundary layers are* ***ultimately turbulent.*** *For a well-shaped and smooth entrance, the boundary layers may be laminar at the upstream end of the zone of non-established flow, followed by turbulent boundary layers in the downstream portion of this region, Fig. 3.24. Between the turbulence (of boundary layers or*

established flow) and solid boundary, there exists the viscous sub layer (to satisfy the conditions of zero velocity on the walls).



***Note:* Comparison of flat plate boundary layers with those of the pipe entrance**

Comparing Fig. 3.9 and 3.24, superficial similarity is noted immediately, however, for pipe entrance, these differences exist:
The plate has been rolled into a cylinder, so it is not flat.
The core velocity steadily increases, while for flat plate case, the free stream velocity remains constant. The pressure diminishment in the fluid in the downstream direction, where for flat plate there is no such pressure variation.

THEORY OF BOUNDARY LAYERS

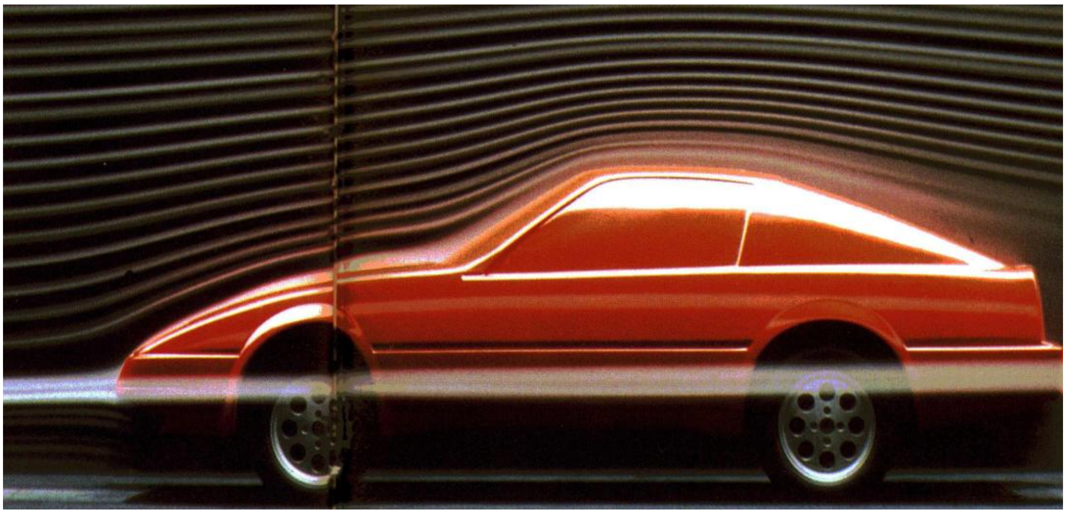
13-1. Introduction

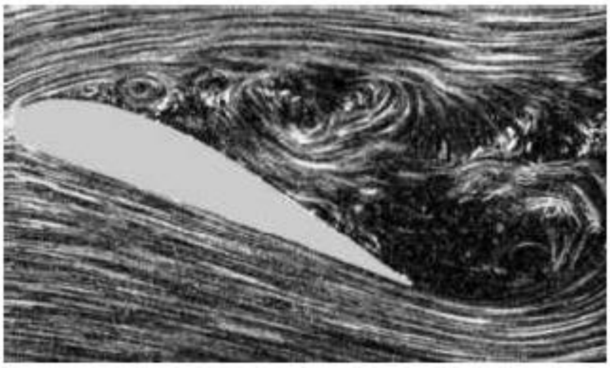
External flows past object encompass an extremely wide variety of fluid mechanics phenomena. Clearly the character of the flow field is a function of the shape of the body. For a given shaped object, the characteristics of the flow depend very strongly on various parameters such as size, orientation, speed, and fluid properties. According to dimensional analysis arguments, the character of the flow should depend on the various dimensionless parameters involved. For typical external flows the most important of these parameters are the Reynolds number $Re = UL/\nu$, where L– is characteristic dimension of the body. For many high-Reynolds-number flows the flow field may be divided into two regions.

1. A viscous boundary layer adjacent to the surface of the vehicle

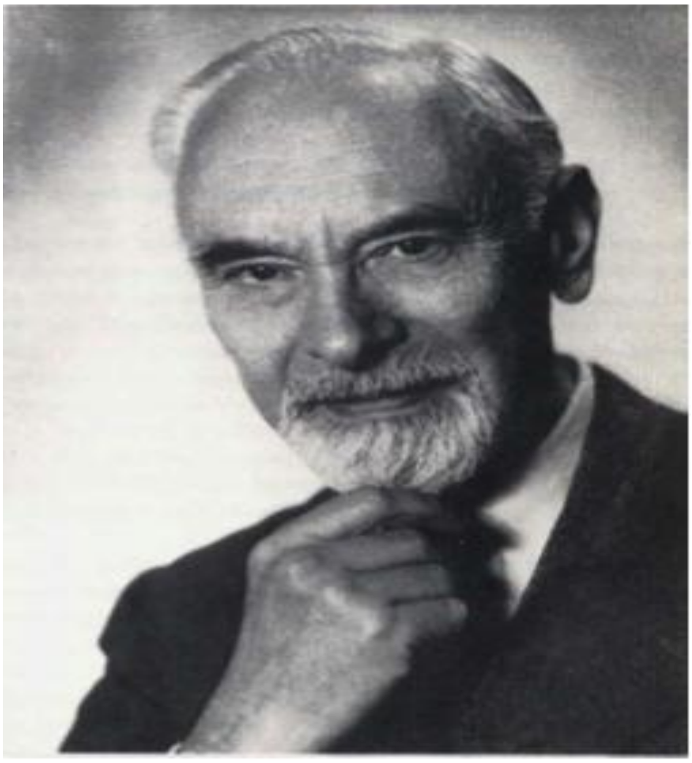
2. The essentially inviscid flow outside the boundary layer

W knows that fluids adhere the solid walls and they take the solid wall velocity. When the wall does not move also the velocity of fluid on the wall is zero. In region near the wall the velocity of fluid particles increases from a value of zero at the wall to the value that corresponds to the external "frictionless" flow outside the boundary layer (see figure).





The concept of boundary layer was first introduced by L. Prandtl in 1904 and since then it has been applied to several fluid flow problems.



Ludwig Prandtl
4.2.1875 – 15.8.1953

When a real fluid (viscous fluid) flows past a stationary solid boundary, a layer of fluid which comes in contact with the boundary surface, adheres to it (on account of viscosity) and condition of no slip occurs (The no-slip condition implies that the velocity of fluid at a solid boundary must be same as that of boundary itself)- Thus the layer of fluid which

cannot slip away from the boundary surface undergoes retardation; this retarded layer further causes retardation for the adjacent layers of the fluid, thereby developing a small region in the immediate vicinity of the boundary surface in which the velocity of the. flowing fluid increases rapidly from zero at the boundary surface and approaches the velocity of main stream. The layer adjacent to the boundary is known as boundary layer. Boundary layer is formed whenever there is relative motion between the boundary and the fluid. Since $\tau_o = \mu \left(\frac{\partial u}{\partial y} \right)_{y=0}$ *, the fluid exerts a shear stress on the boundary and boundary exerts an equal and opposite force on fluid known as the shear resistance.*

According to boundary layer theory the extensive fluid medium around bodies moving in fluids can be divided into following two regions.

(i) a thin layer adjoining the boundary called the boundary layer where the viscous shear takes place.

(ii) A region outside the boundary layer where the flow behavior is quite likes that of an ideal fluid and the potential flow theory is applicable.

1-2. Boundary Layer Definitions and Characteristics

Consider the boundary layer formed on a flat plate kept parallel to flow of fluid of velocity U (Fig. 1-1) (Though the growth of a boundary layer depends upon the body shape, flow over a flat plate aligned in the direction of flow is considered, since most of the flow surfaces can be approximated to a flat plate and for simplicity).

— *The edge facing the direction of flow is called leading edge.*
— *The rear edge is called the trailing edge.*
— *Near the leading edge of a flat plate, the boundary layer is wholly laminar. For a laminar boundary layer the velocity distribution is parabolic.*

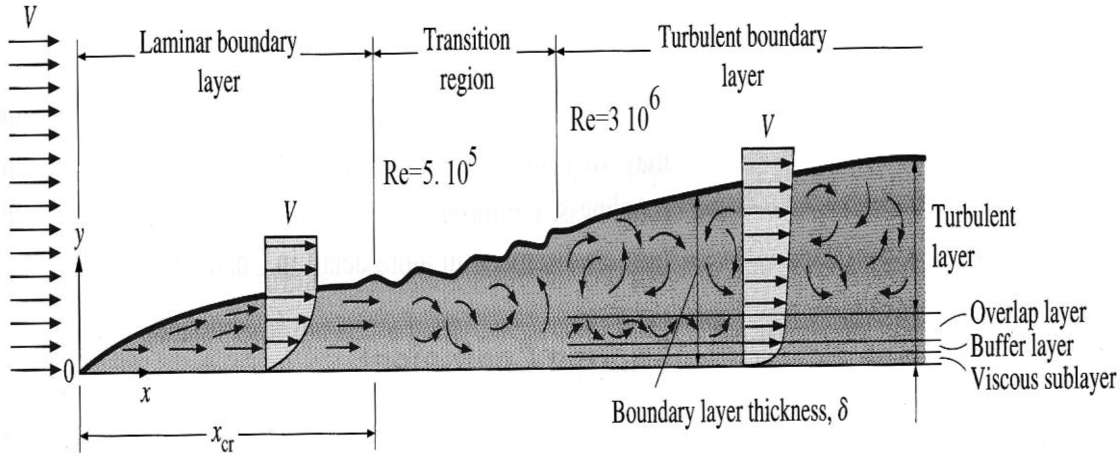


Fig.,1 Boundary layer on a flat plate.

— The thickness of the boundary layer (δ) increases with distance from the leading edge x, as more and more fluid is slowed down by the viscous boundary, becomes unstable and breaks into turbulent boundary layer over a transition region.

For a turbulent boundary layer, if the boundary is smooth, the roughness projections are covered by a very thin layer which remains laminar, called laminar sub layer. The velocity distribution in the turbulent boundary layer is given by Log law or Prandtl's one-seventh power law.

The characteristics of a boundary layer may be summarized as follows:

(i) δ (thickness of boundary layer) increases as distance from leading edge x increases.

(ii) δ decreases as U increases.

(III) δ increases as kinematic viscosity (v) increases.

(iv) $\tau_o = \mu\left(\frac{U}{\delta}\right)$; hence τ_o decreases as x increases. However, when boundary layer becomes turbulent, it shows a sudden increase and then decreases with increasing x.

(v) When U increases in the downward direction, boundary layer growth is reduced.

(vi) When U decreases in the downward direction, flow near the boundary is further retarded, boundary layer growth is faster and is susceptible to separation.

(vii) The various characteristics of the boundary layer on flat plate (e.g. variation of δ, τ_o or force F) are governed by inertial and viscous forces; hence they are functions of either $\frac{UL}{\nu}$ or $\frac{Ux}{\nu}$.

(viii) If $\frac{Ux}{\nu} \leq 5x10^5$ *... boundary layer is laminar (velocity distribution is parabolic).*

If $\frac{Ux}{\nu} \geq 5x10^5$ *... boundary layer is turbulent on that portion (velocity distribution follows Log law or a power law).*

(ix) Critical value of $\frac{Ux}{\nu}$ *at which boundary layer change from laminar to turbulent depends on:*

- *Turbulence in ambient flow,*
- *Surface roughness.*
- *Pressure gradient.*
- *Plate curvature, and*
- *Temperature difference between fluid and boundary.*

(x) Though the velocity distribution would be a parabolic curve in the laminar sub-layer zone, but in view of the very small thickness we can reasonably assume that velocity distribution is linear and so the velocity gradient can be considered constant.

13-2-1. Boundary layer thickness (δ)

The velocity within the boundary layer increases from zero at the boundary surface to the velocity of the main stream asymptotically. Therefore the thickness of the boundary layer is arbitrarily defined as that distance from the boundary in which the velocity reaches 99 per

cent of the velocity of the free stream (u = Q-99U). It is denoted by the symbol δ. This definition however gives an approximate value of the boundary layer thickness and hence δ is generally termed as nominal thickness of the boundary layer.

The boundary layer thickness for greater accuracy is defined in terms of certain mathematical expressions which are the measure of the boundary layer on the flow. The commonly adopted definitions of the boundary layer thickness are:

1. *Displacement thickness (δ^*)*
2. *Momentum thickness (θ)*
3. *Energy thickness (δ_e).*

13-2-2. Displacement thickness (δ^*)

The displacement thickness can be defined as follows:

"It is the distance, measured perpendicular to the boundary, by which the main/free stream is displaced on account of formation of boundary layer."

Or "It is an additional "wall thickness" that would have to be added to compensate for the reduction in flow rate on account of boundary layer formation".

LAILA Page 6 5/9/2015

Figure 2: Definition of boundary layer thickness:
(a) Standard boundary layer (u = 99%U), (b) Boundary layer displacement thickness.

Far-away free stream "sees" solid wall as if it was being displaced up by $\delta^*(x)$ at x.

$$\boxed{U\delta_1{}^* = \int_0^{\delta}(U-u)dy}$$

The displacement thickness is denoted by δ^*.

Let fluid of density ρ flow past a stationary plate with velocity U as shown in the Fig.2. Consider an elementary strip of thickness dy at a distance y from the plate.

Assuming unit width, the mass flow per second through the elementary strip. = ρu.dy Mass flow per second through the elementary strip (unit width) if the plate were not there = ρUdy Reduction of mass flow rate through the elementary strip = ρ(U-u)dy [The difference (U-u) is called velocity of defect].

Total reduction of mass flow rate due to introduction of plate $= \int_0^{\delta} \rho(U-u)dy$

(if the fluid is incompressible). Let the plate is displaced by a distance δ and velocity of flow for the distance δ* is equal to the main/free stream velocity (i.e. U). Then, loss of the mass of the fluid/sec, flowing through the distance δ**

$$= \rho U \delta^*$$

Equating Eqn. (III) and (IV), we get

$$\rho U \delta^* = \int_0^{\delta} (U-u)dy$$

Or $\delta^* = \int_0^{\delta} \left(1 - \frac{u}{U}\right) dy$

1-2-3. Momentum thickness (θ)

Momentum thickness is defined as the distance through which the total loss of momentum per second be equal to if it were passing a stationary plate. It is denoted by θ.

It may also be defined as the distance, measured perpendicular to the boundary of the solid body, by which the boundary should be displaced to compensate for reduction in momentum of the flowing fluid on account of boundary layer formation.

Refer fig. 1.2. Mass of flow per second through the elementary strip $= \rho u dy$

Momentum/sec, of this fluid inside the boundary layer $= \rho u dy x u = \rho u^2 dy$

Momentum/sec, of the same mass of fluid before entering boundary layer $= \rho u U dy$

Loss of momentum/sec. $= \rho u U dy - \rho u^2 dy = \rho u (U-u) dy$

Total loss of momentum/sec.

$$\int_0^\delta \rho u(U-u)dy \qquad \text{...(0}$$

Let θ = distance by which plate is displaced when the fluid is flowing with a constant velocity U.

Then loss of momentum/sec, of fluid flowing through distance θ with a velocity U

$= \rho\theta U^2$*, Equating Eqns. (i) and (ii), we have*

$$\rho\theta U^2 = \int_0^\delta \rho u(U-u)dy\,,\ \theta = \int_0^\delta \frac{u}{U}\left(1-\frac{u}{U}\right)dy$$

All three boundary layer thickness definition δ, δ_1, θ are use in boundary layer analysis.

Viscous/friction effect in a boundary layer causes **mass and momentum flux deficits** (at cross section *x*) relative to ideal inviscid/frictionless flow. If it were not for the viscous effect, the velocity profile would be full/uniform.
In general, the mass and momentum flux deficits increase with *x* and the boundary layer grow with *x*.
1. Momentum flux deficit increases because the fluid in the boundary layer is continuously subjected to opposing (net) shear as the BL develops along the wall. Here, we focus mainly in the case of no pressure gradient (uniform pressure).
- If a fluid element within a BL is considered, the net shear is illustrated in Fig. 2.
- if a semi-infinite CV is considered, the net shear is the wall shear stress as illustrated in Figs. 2 and 8.
2. Recall that the concept of flux is associated with a surface, i.e., flux through a surface. Here the surface is a semi-infinite plane ($0 < y \to \infty$) at *x*.]

Motivation: Due to the force/mechanical interaction between the body and the fluid stream and Newton's 3rd law of motion, the lost of momentum of fluid stream must relate to force on the body.
Question: In order to study the BL a little more conveniently, how should we **parameterize** (/describe) the deficit in mass and momentum fluxes at any cross section *x*?

Viscous/friction effect in a boundary layer causes fluid in the boundary layer to be retarded, losing its velocity/momentum, and resulting in **mass flux deficit relative to ideal inviscid/frictionless flow**. If it were not for the viscous effect, the velocity profile would be full/uniform.

Momentum flux deficit of the actual mass flux: *dM& x,deficit* = (*U* − *u*)(ρ*udA*). [Note that the local free stream velocity is used as a reference/datum.]

Viscous/friction effect in a boundary layer causes fluid in the boundary layer to be retarded, losing its velocity/momentum, and resulting in **momentum flux deficit relative to ideal inviscid/frictionless flow**. If it were not for the viscous effect, the velocity profile would be full/uniform.

Note that we account for the momentum flux deficit ($U - u$) of the actual mass flux $\rho u dA$, not of an ideal mass flux $\rho U dA$. [We want to know the lost of momentum of the actual mass.]

Comments on Thicknesses: Mathematical Definitions and Physical Meanings

In "deriving/arriving at" the "definitions"1 of the displacement and momentum thicknesses given by Eqn. (1) and (2), respectively, we have (implicitly or explicitly) restricted ourselves to, say, incompressible flow, *in order to* gain some *physical meanings* of the thicknesses *for such flow*; namely, Definitions Physical Meanings Displacement thickness - related to - Mass flux deficit.

Momentum thickness – related to - Momentum flux deficit.

The question may naturally arise whether these thicknesses "can be used" for, say, compressible flow. In this regard, there are couples of points to consider:

1. If we want to *preserve* the definitions:

These definitions Eqns. (1) and (2), can simply be taken as *definitions*, hence, the "arriving/deriving" that was done is irrelevant and not necessary – one need not do it in order to so *define* "δ^* " and "θ ." [However, one would probably and initially then not know what it was for or represented.] If so desired, both "δ^*" and "θ" can be calculated for compressible flows.

[At any instant in time *t* and at any cross section *x*, one can evaluate the integrals *regardless* of whether it is incompressible or compressible flow.]

The questions are then rather:

○ Whether they preserve the physical meanings (still have the same physical meanings) as in the case of incompressible flow, i.e., δ^* corresponds to mass flux deficit and θ corresponds to momentum flux deficits?

○ If they do not, what physics do they represent/mean in compressible flow?

2. If we want to *preserve* the physics/physical meanings:

On the other hand, if we want to preserve the physical meanings (if the two previously defined definition do not preserve the physical meanings), we have to *redefine* "δ^*" and "θ" accordingly. New definitions – preferably with wider scope - must be sought to represent the mass and momentum flux deficits in compressible flow (and preferably still applicable also to incompressible flow).

Question:

Preserve its physical meaning in compressible flow? That is, does still represent mass flux deficit. If not:

- What physics does it represent? And - How should we redefine δ^* in order to preserve the physical meaning of mass flux deficit? What about

1-2-4. Energy thickness (δ_e)

Energy thickness is defined as the distance, measured perpendicular to the boundary of the solid body, by which the boundary should be displaced to compensate for the reduction in K.E. of the flowing fluid on account of boundary layer formation. It is denoted by δ_e.

Refer fig.1.2

Mass of flow per second through the elementary strip $= \rho u dy$

K.E. of this fluid inside the boundary layer

$$= \frac{1}{2} m u^2 = \frac{1}{2}(\rho u dy) u^2$$

K.E. of the same mass of fluid before entering the boundary layer

$$= \frac{1}{2}(\rho u dy) U^2$$

Loss of K.E. through elementary strip

$$= \frac{1}{2}(\rho u dy) U^2 - \frac{1}{2}(\rho u dy) u^2 = \frac{1}{2}\rho u\left(U^2 - u^2\right) dy$$

$\therefore$ *Total loss of K.E. of fluid* $\int_0^\delta \frac{1}{2}\rho u\left(U^2 - u^2\right) dy$

Let δ_e = distance by which the plate is displaced to compensate for the reduction in K.E. Then loss of K.E. through δ_e of fluid flowing with velocity U $\frac{1}{2}(\rho U \delta_e) U^2$

Equating eqns. (i) and (ii). We have

$$\frac{1}{2}(\rho U \delta_e) U^2 = \int_0^\delta \frac{1}{2}\rho u\left(U^2 - u^2\right) dy$$

$$\delta_e = \frac{1}{U^3}\int_0^\delta u\left(U^2 - u^2\right) dy = \int_0^\delta \frac{u}{U}\left(1 - \frac{u^2}{U^2}\right) dy$$

Example 1.1.

The velocity distribution in the boundary layer is given by: $\frac{u}{U}=\frac{y}{\delta}$ *, where u is the velocity at a distance y from the plate and u = U at y =δ, δ being boundary layer thickness. Find:*

(i) The displacement thickness, *(ii) the momentum thickness,*

(iii) The energy thickness, and *(iv) the value of* $\frac{\delta^*}{\theta}$

Sol. Velocity distribution: $\frac{u}{U}=\frac{y}{\delta}$ ……………………………… *(Given)*

(i) The displacement thickness, δ^**:* $\delta^*=\int_0^\delta\left(1-\frac{u}{U}\right)dy$

$$\delta^*=\int_0^\delta\left(1-\frac{y}{\delta}\right)dy \quad \because \frac{u}{U}=\frac{y}{\delta}$$

$$=\left[y-\frac{y^2}{2\delta}\right]_0^\delta$$

$$\delta^*=\left(\delta-\frac{\delta^2}{2\delta}\right)=\delta-\frac{\delta}{2}=\frac{\delta}{2} \qquad \text{(Ans.)}$$

(ii) The momentum thickness, θ*:*

$$\theta=\int_0^\delta\frac{u}{U}\left(1-\frac{u}{U}\right)dy$$

$$\theta=\int_0^\delta\frac{y}{\delta}\left(1-\frac{y}{\delta}\right)dy=\int_0^\delta\left(\frac{y}{\delta}-\frac{y^2}{\delta^2}\right)dy$$

$$\theta == \left(\frac{y^2}{2\delta} - \frac{y^3}{3\delta^2}\right)_0^\delta = \frac{\delta^2}{2\delta} - \frac{\delta^3}{3\delta^2} = \frac{\delta}{6} \qquad (Ans.)$$

(iii) The energy thickness, δ_e:

$$\delta_e = \int_0^\delta \frac{u}{U}\left(1 - \frac{u^2}{U^2}\right)dy$$

$$\delta_e = \int_0^\delta \frac{y}{\delta}\left(1 - \frac{y^2}{\delta^2}\right)dy = \int_0^\delta \left(\frac{y}{\delta} - \frac{y^3}{\delta^3}\right)dy$$

$$\left[\frac{y^2}{2\delta} - \frac{y^4}{4\delta^3}\right]_0^\delta = \frac{\delta^2}{2\delta} - \frac{\delta^4}{4\delta^3} = \frac{\delta}{4}$$

$$\therefore \delta_e = \frac{\delta}{4}$$

(iv) The value of $\frac{\delta^*}{\theta}$:

$$\frac{\delta^*}{\theta} = \frac{\delta/2}{\delta/6} = 3.0 \quad \text{(Ans.)}$$

__Example.1-2.__

The velocity distribution in the boundary layer is given by $\frac{u}{U} = \frac{3}{2}\frac{y}{\delta} - \frac{1}{2}\frac{y^2}{\delta^2}$, δ being boundary layer thickness.

Calculate the following:

(i) The ratio of displacement thickness to boundary layer thickness $\frac{\delta^*}{\delta}$.

(ii) The ratio of momentum thickness to boundary layer thickness $\frac{\theta}{\delta}$.

Sol. Velocity distribution: $\frac{u}{U} = \frac{3}{2}\frac{y}{\delta} - \frac{1}{2}\frac{y^2}{\delta^2}$ *(given)*

(i) $\frac{\delta^*}{\delta}$:

$$\delta^* = \int_0^\delta \left(1 - \frac{u}{U}\right) dy = \int_0^\delta \left(1 - \frac{3}{2}\frac{y}{\delta} + \frac{1}{2}\frac{y^2}{\delta^2}\right) dy$$

$$= \left[y - \frac{3}{2}\frac{y^2}{2\delta} + \frac{1}{2}\frac{y^3}{3\delta^2} \right]_0^\delta$$

$$= \left[\delta - \frac{3}{4}\frac{\delta^2}{\delta} + \frac{1}{2}\frac{\delta^3}{3\delta^2} \right] = \left(\delta - \frac{3}{4}\delta + \frac{\delta}{6} \right) = \frac{5\delta}{12}$$

$$\therefore \qquad \frac{\delta^*}{\delta} = \frac{5}{12} \qquad \text{(Ans.)}$$

(ii) $\frac{\theta}{\delta}$:

$$\theta = \int_0^\delta \frac{u}{U}\left(1 - \frac{u}{U}\right) dy$$

$$\theta = \int_0^\delta \left(\frac{3}{2}\frac{y}{\delta} - \frac{1}{2}\frac{y^2}{\delta^2}\right)\left(1 - \frac{3}{2}\frac{y}{\delta} + \frac{1}{2}\frac{y^2}{\delta^2}\right) dy$$

$$\theta = \int_0^\delta \left(\frac{3}{2}\frac{y}{\delta} - \frac{9}{4}\frac{y^2}{\delta^2} + \frac{3}{4}\frac{y^3}{\delta^3} - \frac{1}{2}\frac{y^2}{\delta^2} + \frac{3}{4}\frac{y^3}{\delta^3} - \frac{1}{4}\frac{y^4}{\delta^4}\right) dy$$

$$\theta = \int_0^\delta \left(\frac{3}{2}\frac{y}{\delta} - \left(\frac{9}{4}\frac{y^2}{\delta^2} + \frac{1}{2}\frac{y^2}{\delta^2}\right) + \left(\frac{3}{4}\frac{y^3}{\delta^3} + \frac{3}{4}\frac{y^3}{\delta^3}\right) - \frac{1}{4}\frac{y^4}{\delta^4}\right) dy$$

$$\theta = \int_0^\delta \left(\frac{3}{2}\frac{y}{\delta} - \frac{11}{4}\frac{y^2}{\delta^2} + \frac{3}{2}\frac{y^3}{\delta^3} - \frac{1}{4}\frac{y^4}{\delta^4}\right) dy$$

$$\theta = \left[\frac{3}{2} x \frac{y^2}{2\delta} - \frac{11}{4} x \frac{y^3}{3\delta^2} + \frac{3}{2} x \frac{y^4}{4\delta^3} - \frac{1}{4} x \frac{y^5}{5\delta^4}\right]_0^\delta$$

$$\theta = \left[\frac{3}{2} x \frac{\delta^2}{2\delta} - \frac{11}{4} x \frac{\delta^3}{3\delta^2} + \frac{3}{2} x \frac{\delta^4}{4\delta^3} - \frac{1}{4} x \frac{\delta^5}{5\delta^4}\right]_0^\delta$$

$$\theta = \frac{19}{120}\delta$$

$$\therefore \qquad \frac{\theta}{\delta} = \frac{19}{120}$$

Example.1.6.

Explain what you understand by boundary layer thickness and displacement thickness.

Determine the relationship between the two for a boundary layer which is (i) laminar throughout and (U) turbulent throughout.

Assume (1) in the laminar boundary layer the flow obeys the law, shear stress $\tau = \mu \frac{du}{dy}$ *, where μ the viscosity, which leads to the velocity profile* $(U-u) = k(\delta - y)^2$ *, where U is the free stream velocity, u is the velocity at a distance y above the plate and k is a constant;*

(2) The velocity distribution in the turbulent boundary layer is given $\frac{u}{U} = \left(\frac{y}{\delta}\right)^{\frac{1}{7}}$.

Sol. (a) Boundary layer thickness: It is defined as the distance from the boundary of solid body measured in Y-direction to the point, where the velocity of fluid is approximately equal to 0.99 times the free stream (U) velocity of fluid.

Displacement thickness: It is defined as the distance, measured perpendicular to the boundary of the solid body by which the boundary should be displaced to compensate for the reduction in flow rate on account of boundary layer formation.

(i) When the flow is laminar throughout:

Velocity profile: $(U-u)=k(\delta-y)^2$ *(A)*

where,

U = free stream velocity,

u = velocity at a distance y above plate, and

k = constant.

Let δ = displacement thickness, and*

δ = boundary layer thickness.

$$\delta^* = \int_0^\delta \left(1-\frac{u}{U}\right)dy$$

Dividing Eqn. (A) by U, we get

$$1-\frac{u}{U}=\frac{k}{U}(\delta-y)^2$$

$$\therefore \quad \int_0^\delta \frac{k}{U}(\delta-y)^2 dy = \left[-\frac{k}{U}\,x\,\frac{(\delta - y)^3}{3}\right]_0^\delta = \left[\frac{k(\delta-y)^3}{3U}\right]_0^\delta = \frac{k\delta^3}{3U} \quad \ldots\ldots\ldots \text{(B)}$$

When $y=0, \quad u=0$ i.e. on the surface of the plate.

Substituting these parameters in Eqn. (A), we get $U=k\delta^2$ ……… ……*(C)*

Substituting the value of U from (C) in (B), we get $\delta^* = \dfrac{k\delta^3}{3k\delta^2}=\dfrac{\delta}{3}$ (Ans.)

(ii) When the flow is turbulent throughout:

Velocity profile: $\dfrac{u}{U}=\left(\dfrac{y}{\delta}\right)^{\frac{1}{7}}$.

$$\delta^* = \int_0^\delta \left(1-\frac{u}{U}\right)dy = \int_0^\delta \left[1-\left(\frac{y}{\delta}\right)^{\frac{1}{7}}\right]dy$$

$$\left[y - \frac{7}{8} x \frac{y^{\frac{8}{7}}}{\delta^{\frac{1}{7}}} \right]_0^{\delta} = \frac{1}{8}\delta \qquad \text{(Ans)}$$

Example.1.8.

The velocity distribution in the boundary layer over the face of a spillway was observed to be:

$$\frac{u}{U} = \left(\frac{y}{\delta}\right)^{0.22}$$

The free stream velocity U is 20 m/s and boundary layer thickness 5 cm at a certain section. The discharge is 5 m^3 /s per meter length of spillway. Calculate displacement thickness, energy thickness and loss of energy up to section under consideration.

Sol. Given: Velocity distribution: $\frac{u}{U} = \left(\frac{y}{\delta}\right)^{0.22}$

U = 20 m/s; 6 = 5 cm; Q = m^3/s per meter length of spillway,

$\delta^*, \delta_e, \mathrm{E_L}$:

The displacement thickness is given by the equation:

$$\delta^* = \int_0^{\delta}\left(1 - \frac{u}{U}\right)dy = \int_0^{\delta}\left[1 - \left(\frac{y}{\delta}\right)^{0.22}\right]dy = \left[y - \frac{1}{(\delta)^{0.22}} x \frac{(y)^{0.22+1}}{1.22}\right]_0^{\delta}$$

$$= \delta - \frac{\delta}{1.22} = \frac{0.22}{1.22}\delta = \frac{0.22}{1.22} x5 = 0.9016\,cm \qquad \text{(Ans.)}$$

The energy thickness is given by the equation:

$$\delta_e = \int_0^\delta \frac{u}{U}\left[1-\left(\frac{u}{U}\right)^2\right] = \int_0^\delta \left[\frac{u}{U}-\left(\frac{u}{U}\right)^3\right]dy = \int_0^\delta \left[\left(\frac{y}{\delta}\right)^{0.22} - \left(\frac{y}{\delta}\right)^{0.66}\right]dy$$

$$\delta_e = \int_0^\delta \left[\frac{1}{(\delta)^{0.22}}\, x\, \frac{(y)^{0.22+1}}{1.22} - \frac{1}{(\delta)^{0.66}}\, x\, \frac{(y)^{0.66+1}}{1.66}\right]_0^\delta$$

$$\delta_e = \frac{8}{1.22} - \frac{\delta}{1.66} = \delta\left[\frac{1}{1.22} - \frac{1}{1.66}\right] = 5\left[\frac{1.66-1.22}{1.22x1.66}\right] = 1.086\,\text{cm} \qquad \text{(Ans.)}$$

The energy loss per m length of spillway is:

$$E_L = \frac{1}{2}(\rho\, x\delta_e xU)xU^2 = \frac{1}{2}\rho\delta_e U^3$$

$$E_L = \frac{1}{2}x1000x\left(\frac{1.086}{100}\right)x(20)^3 x10^{-3}\ \text{kNm/s} = 43.44\,\text{kNm/s}$$

Energy loss in terms of m of head:

$$= \frac{E_L}{\rho g Q} = \frac{43.44x1000}{1000x5x9081} = 0.8856m \qquad \text{(Ans.)}$$

Example.1-9. for steady Poiseuille flow in a pipe of radius R, obtain an expression for ratio of the displacement thickness (δ) to momentum thickness (θ).*

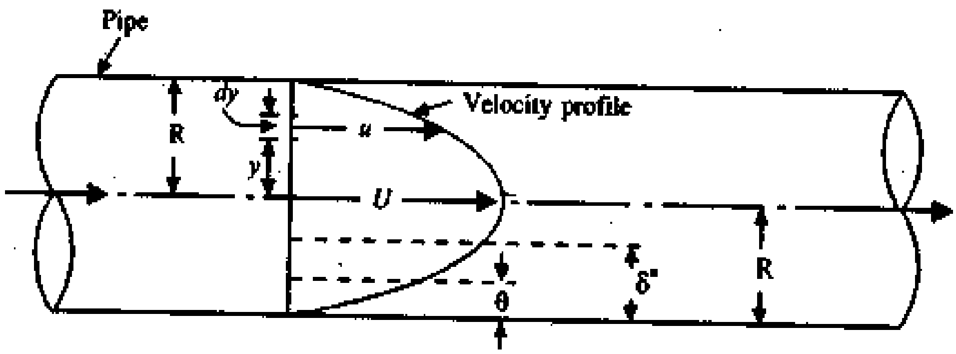


Fig.1.3

Sol. Radius of the pipe = R

For steady Poiseuille flow in a circular pipe, the velocity distribution is given by,

$$u = -\frac{1}{4\mu} \cdot \frac{\partial p}{\partial x} (R^2 - y^2) \ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots (i)$$

Where y being measured from the center of the pipe.

At y=0, u=U (maximum Velocity) $\therefore \quad U = -\frac{1}{4\mu} \cdot \frac{\partial p}{\partial x} R^2 \ldots\ldots\ldots\ldots (ii)$

From the definition of displacement thickness, we have

$$\left[\pi R^2 - \pi\left(R - \delta^*\right)^2\right]U = \int_0^R 2\pi y (U - u) dy$$

or $$2\pi R \delta^* U = \int_0^R 2\pi y (U - u) dy$$

[Neglecting the term containing $(\delta^*)^2$, since δ^* is very small.]

or $$\delta^* = \frac{1}{R}\int_0^R \left(1 - \frac{u}{U}\right) y dy = \frac{1}{R}\int_0^R \frac{y^2}{R^2} y dy = \frac{R}{4}$$

Dividing (i) by (ii), we have, $\frac{u}{U} = \frac{R^2 - y^2}{R^2} = 1 - \frac{y^2}{R^2}$ or $\left(1 - \frac{u}{U}\right) = \frac{y^2}{R^2}$

From the definition of momentum thickness, we have

$$\left[\pi R^2 - \pi(R - \theta)^2\right]\rho U = \int_0^R 2\rho \pi \, y.dy.u(U - u)$$

On simplification, we get

$$2\pi R \theta U^2 = \int_0^R 2\pi y u (U - u) dy$$

$$\theta = \frac{1}{R}\int_0^R \frac{u}{U}\left(1-\frac{u}{U}\right) y dy$$

$$\theta = \frac{1}{R}\int_0^R \left(1-\frac{y^2}{R^2}\right)\frac{y^2}{R^2} y dy = \frac{1}{R}\int_0^R \left(\frac{y^3}{R^2}-\frac{y^5}{R^4}\right) dy$$

$$\theta = \frac{1}{R}\left[\frac{y^4}{4R^2}-\frac{y^6}{6R^4}\right]_0^R = \frac{R}{12}$$

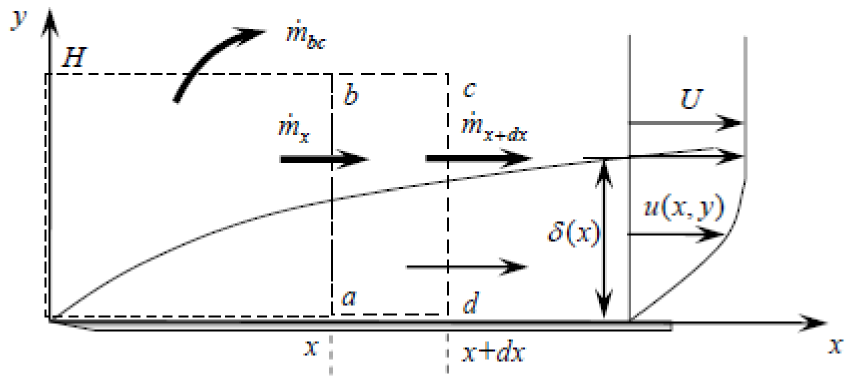
$$\frac{\delta^*}{\theta} = \frac{R/4}{R/12} = 3 \ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots(Ans.)$$

13-3. Momentum Equation for Boundary Layer by Vonkàrman

Vonkàrman suggested a method based on the momentum equation by the use of which the growth of a boundary layer along a flat plate, the wall shear stress and the drag force could be determined (when the velocity distribution in the boundary layer is known). Starting from the beginning of the plate, the method can be used for both laminar and turbulent boundary layers.

Fig.1-4. shows a fluid flowing over a thin plate (placed at zero incidence) with a free stream velocity equal to U. Consider a small length dx of the plate at a distance x from the leading edge as shown in Fig. 13-4 (a); the enlarged view of the small length of the plate is

Mass diagram:



X-Momentum diagram:

In this diagram, we shall include the non-uniform pressure force. However, in the derivation below we shall consider only the case of uniform pressure. For the case of non-uniform pressure, we shall postpone it to later section.

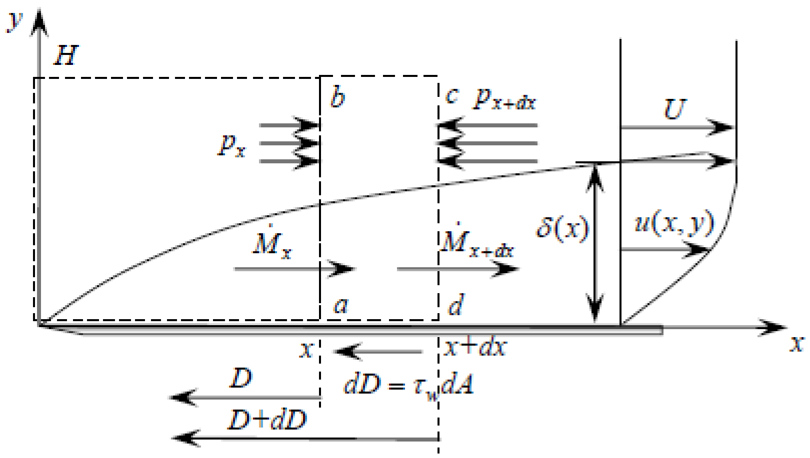


Fig.1-4.Mass and momentum diagrams for the derivation of Vonkàrman's momentum integral equation

Consider unit width of plate perpendicular to the direction of flow.

Let ABCD be a small element of a boundary layer (the edge BC represents the outer edge of the boundary layer).

Mass rate of fluid entering through AB

$$= \int_0^{\delta} \rho u dy$$

Mass rate of fluid leaving through CD

$$= \int_0^{\delta} \rho u dy + \frac{d}{dx}\left[\int_0^{\delta} \rho u dy\right] dx$$

$\therefore$ *Mass rate of fluid entering the control volume through the surface AB = mass rate of fluid through BC – mass rate of fluid through CD*

$$= \int_0^\delta \rho u dy + \frac{d}{dx}\left[\int_0^\delta \rho u dy\right]dx - \int_0^\delta \rho u dy = \frac{d}{dx}\left[\int_0^\delta \rho u dy\right]dx$$

The fluid is leaving through BC with a uniform velocity U.

Momentum rate of fluid entering the control volume in X-direction through AB

$$= \int_0^\delta \rho u^2 dy$$

Momentum rate of fluid leaving the control volume in X-direction through CD

$$= \int_0^\delta \rho u^2 dy + \frac{d}{dx}\left[\int_0^\delta \rho u^2 dy\right]dx$$

Momentum rate of fluid entering the control volume through BC in X-direction

$$= \frac{d}{dx}\left[\int_0^\delta \rho u dy\right]dx \,\mathrm{x}\, U = \frac{d}{dx}\left[\int_0^\delta \rho u U dy\right]dx$$

$\therefore$ *Rate of change of momentum of control volume = momentum rate of fluid through AB – momentum rate of fluid through BC – momentum rate of fluid through CD*

$$= \int_0^\delta \rho u^2 dy + \frac{d}{dx}\left[\int_0^\delta \rho u^2 dy\right]dx - \int_0^\delta \rho u^2 dy - \frac{d}{dx}\left[\int_0^\delta \rho u U dy\right]dx$$

$$= \frac{d}{dx}\left[\int_0^\delta \rho u^2 dy - \int_0^\delta \rho u U dy\right]dx$$

$$= \frac{d}{dx}\left[\int_0^\delta \left(\rho u^2 dy - \rho u U dy\right)\right]dx$$

$$= \frac{d}{dx}\left[\rho\int_0^\delta \left(u^2 - uU\right)dy\right]dx$$

$(\rho \text{ is constant for incompressible fluid})$

$$= \rho\frac{d}{dx}\left[\int_0^\delta \left(u^2 - uU\right)dy\right]dx$$

As per momentum principle the rate of change of momentum on the control volume ABCD must be equal to the total force on the control volume in the same direction. The only external force acting on the control volume is the shear force acting on the side AD in the direction D to A (fig. 1-4 b). The value of this force (drag force) is given by.

$\Delta F_D = \tau_o \mathrm{x} dx$

Thus the total external force in the direction of rate of change of momentum:

$= -\tau_o \mathrm{x} dx$

Equating the equations (1-4) and (1-5), one has

$$-\tau_o \mathrm{x} dx = \rho \frac{d}{dx}\left[\int_0^\delta \left(u^2 - uU\right)dy\right]dx$$

$$\tau_o = -\rho \frac{d}{dx}\left[\int_0^\delta \left(u^2 - uU\right)dy\right]$$

$$\tau_o = \rho \frac{d}{dx}\left[\int_0^\delta \left(uU - u^2\right)dy\right]$$

$$\tau_o = \rho \frac{d}{dx}\left[\int_0^\delta U^2\left(\frac{u}{U} - \frac{u^2}{U^2}\right)dy\right]$$

$$\tau_o = \rho U^2 \frac{d}{dx}\left[\int_0^\delta \frac{u}{U}\left(1 - \frac{u}{U}\right)dy\right]$$

$$\frac{\tau_o}{\rho U^2} = \frac{d}{dx}\left[\int_0^\delta \frac{u}{U}\left(1 - \frac{u}{U}\right)dy\right]$$

$$But \ \int_0^\delta \frac{u}{U}\left(1 - \frac{u}{U}\right)dy = momentum\ thickness\,(\theta)$$

$$\therefore \qquad \frac{\tau_o}{\rho U^2} = \frac{d\theta}{dx}$$

Eqn. (1.7) is known as:
Von Karman momentum equation for boundary layer flow, and is used to find out the frictional drag on smooth flat plate for both laminar and turbulent boundary layer.
The following boundary conditions must be satisfied for any assumed velocity distribution:

(i) T the surface of the plate: $y = 0, \ \ \mathrm{u} = 0, \ \ \frac{\mathrm{du}}{dy} = \text{finite value}$

(ii) At the outer edge of the boundary layer: $y = \delta, \ \ \mathrm{u} = \mathrm{U}$,

$$y = \delta, \ \ \frac{\mathrm{du}}{\mathrm{dy}} = 0$$

The shear stress for a given velocity profile in laminar, transition or turbulent zone is obtained from equation (1-6) or (1-7). Then drag force on a small distance dx of a plate is given by,
$\Delta F_D = shear$ stress x area $= \tau_o \mathrm{x}(\mathrm{B} \text{ x } dx)$ *[assuming width of the plate as unity], where B = width of the plate)*
$\therefore$ *Total drag on the plate of length L one side,*

$$\Delta F_D = \int \Delta F_D = \int_0^L \tau_o \mathrm{xBx}\, dx$$

The ratio of the shear stress $\tau_o = \frac{1}{2}\rho U^2$*, is known as "Local coefficient of drag" (or coefficient of skin friction) and is denoted by* C_D^*

i.e. $$C_D^* = \frac{\tau_o}{\frac{1}{2}\rho U^2}$$

- The ratio of the drag force to the quantity $\frac{1}{2}\rho A U^2$ *is called "Average coefficient of drag" and is denoted by* C_D

i.e. $$C_D = \frac{F_D}{\frac{1}{2}\rho A U^2}$$

Where, ρ *= mass density of fluid,*
A = area of surface/ plate, and
U = free stream velocity

NOTE: It is interesting to note here that, from Eqn. (7), the increase in drag force dD is directly proportional to the increase in the momentum thickness $d\theta$*. That is,*

we can then summarize the von Kármán's momentum integral equation for a boundary layer in uniform pressure as follows:

von Kármán's Momentum Integral Equation:

$$\tau_w \left(= \frac{1}{w} \frac{dD}{dx} \right) = \rho U^2 \frac{d\theta}{dx}, \qquad \frac{C_f}{2} = \frac{d\theta}{dx}$$

$$dD = w\rho U^2 d\theta, \qquad D(x_1, x_2) = w\rho U^2 [\theta(x_2) - \theta(x_1)]$$

Note: In fluid, the *dimensionless form* is often preferred owing partly to the concept of measuring things with its own scales which often gives insight into the problem as well as the *relative order of magnitude* of various terms. For example, in a wide range of flat plate flows, $C_f \sim 10^{-5} - 10^{-2}$. This means that, the wall shear stress τ_w is relatively much less than the dynamic pressure $(1/2)\rho U^2$ in the flow, e.g., $0.001 - 1\%$. Certainly, it can be a little cumbersome in some situations, e.g., calculate the magnitude of τ_w in Pa, etc.

As we can see, the momentum thickness so defined from the momentum flux deficit relates directly to the drag force.

1-4. Laminar Boundary layer:

Let us find out boundary layer thickness (δ), shear stress (τ_o), local coefficient of drag (C_D^*) and coefficient of drag (C_D) for the following velocity distribution in the laminar boundary layer:

1. $\frac{u}{U} = 2\left(\frac{y}{\delta}\right) - \left(\frac{y}{\delta}\right)^2$

2. $\frac{u}{U} = \frac{2}{3}\left(\frac{y}{\delta}\right) - \frac{1}{2}\left(\frac{y}{\delta}\right)^3$ ……………….. *Prandtl's velocity distribution.*

3. $\frac{u}{U} = 2\left(\frac{y}{\delta}\right) - 2\left(\frac{y}{\delta}\right)^3 + \left(\frac{y}{\delta}\right)^4$

4. $\frac{u}{U} = \sin\left(\frac{\pi}{2}\frac{y}{\delta}\right)$

Case1. Velocity distribution: $\frac{u}{U} = 2\left(\frac{y}{\delta}\right) - \left(\frac{y}{\delta}\right)^2$

(i) Boundary layer thickness:

We know,

$$\frac{\tau_o}{\rho U^2} = \frac{d}{dx}\left[\int_0^\delta \frac{u}{U}\left(1 - \frac{u}{U}\right)dy\right]$$

Substituting the value of u/U, we get

$$\frac{\tau_o}{\rho U^2} = \frac{d}{dx}\left[\int_0^\delta \left(\frac{2y}{\delta} - \frac{y^2}{\delta^2}\right)\left[1 - \left(\frac{2y}{\delta} - \frac{y^2}{\delta^2}\right)\right]dy\right]$$

$$\frac{\tau_o}{\rho U^2} = \frac{d}{dx}\left[\int_0^\delta \left(\frac{2y}{\delta} - \frac{4y^2}{\delta^2} + \frac{2y^3}{\delta^3} - \frac{y^2}{\delta^2} + \frac{2y^3}{\delta^3} - \frac{y^4}{\delta^4}\right)dy\right]$$

$$\frac{\tau_o}{\rho U^2} = \frac{d}{dx}\left[\int_0^\delta \left(\frac{2y}{\delta} - \frac{5y^2}{\delta^2} + \frac{4y^3}{\delta^3} - \frac{y^4}{\delta^4}\right)dy\right]$$

$$\frac{\tau_o}{\rho U^2} = \frac{d}{dx}\left[\left(\frac{2y^2}{2\delta} - \frac{5y^3}{3\delta^2} + \frac{4y^4}{4\delta^3} - \frac{y^5}{5\delta^4}\right)\right]_0^\delta$$

$$\frac{\tau_o}{\rho U^2} = \frac{d}{dx}\left(\delta - \frac{5}{3}\delta + \delta - \frac{1}{5}\delta\right) = \frac{d}{dx}\left(\frac{2}{15}\delta\right)$$

$$\tau_o = \rho U^2 \mathrm{x} \frac{d}{dx}\left(\frac{2}{15}\delta\right) = \frac{2}{15}\rho U^2 \frac{d\delta}{dx}$$

Also, according to Newton's low of viscosity,

$$\tau_o = \mu\left(\frac{du}{dy}\right)_{y=0}$$

But $u = U\left(\frac{2y}{\delta} - \frac{y^2}{\delta^2}\right)$

And $\frac{du}{dy} = U\left(\frac{2}{\delta} - \frac{2y}{\delta^2}\right)$, U being costant

$$\left(\frac{du}{dy}\right)_{y=0} = U\left(\frac{2}{\delta} - 0\right) = \frac{2U}{\delta}$$

Substituting this value in (ii), we get

$$\tau_o = \frac{2\mu U}{\delta}$$

Equating the values of τ_o *, given by Eqn (1.11) and (1.12), we get*

$$\frac{2}{15}\rho U^2 \frac{d\delta}{dx} = \frac{2\mu U}{\delta} .$$

$$\delta \frac{d\delta}{dx} = \frac{15\mu U}{\rho U^2} = \frac{15\mu}{\rho U}$$

$$\delta . d\delta = \frac{15\mu}{\rho U} dx$$

Integrating both sides, we get

$\frac{\delta^2}{2} = \frac{15\mu}{\rho U} x + C$ *,(where C = constant of integration)*

At $x = 0,\ \delta = 0 \quad \therefore\ C = 0$

$$\frac{\delta^2}{2} = \frac{15\mu x}{\rho U} \qquad \therefore\quad \delta = \sqrt{\frac{2x15\mu \mathbf{x}}{\rho U}} = 5.48\sqrt{\frac{\mu x}{\rho U}}$$

$$\therefore\quad \delta = 5.48\sqrt{\frac{\mu \mathbf{x}\ x\ x}{\rho U x x}} = 5.48\sqrt{\frac{x^2}{Re_x}}$$

Where $Re_x = \dfrac{\rho U x}{\mu}$ or $\delta = 5.48\dfrac{x}{\sqrt{Re_x}}$

(ii) Shear stress, τ_o*:*

From Eqn. (1.12), we have

$$\tau_o = \frac{2\mu U}{\delta} \qquad \textit{But} \qquad \delta = 5.48\frac{x}{\sqrt{Re_x}}$$

$$\therefore \tau_o = \frac{2\mu U}{5.48\dfrac{x}{\sqrt{Re_x}}} = \frac{2\mu U\sqrt{Re_x}}{5.48x} = 0.365\frac{\mu U}{x}\sqrt{Re_x}\ ....(1.14)$$

(iii) Local co-efficient of drag, C_D^**:*

$$\because \tau_o = 0.365\frac{\mu U}{x}\sqrt{Re_x}$$

$$\therefore \tau_o = C_D^*\frac{\rho U^2}{2}$$

(Where $C_D^* =$ *local co-efficient of drag)*

Equating the two values of τ_o*, given by equation (1.14) and (1.9) we get*

$$C_D^* = \frac{\left[0.365\dfrac{\mu U}{x}\sqrt{\mathrm{Re}_x}\right]}{\dfrac{\rho U^2}{2}} \qquad \text{hence } C_D^* = \frac{0.73}{\sqrt{\mathrm{Re}_x}}$$

(iv)Coefficient of drag, C_D:

We know that, $C_D = \dfrac{F_D}{\frac{1}{2}\rho A U^2}$

Where $F_D = \int_0^L \tau_O \text{ x B x d}x$

$$F_D = \int_0^L \left[0.365\frac{\mu U}{x}\sqrt{\mathrm{Re}_x}\right]\text{x B x d}x$$

$$= \int_0^L \left[0.365\frac{\mu U}{x}\sqrt{\frac{\rho U x}{\mu}}\right]\text{x B x d}x \quad \left(\because \mathrm{Re}_x = \frac{\rho U x}{\mu}\right)$$

$$= 0.365\int_0^L \mu U\sqrt{\frac{\rho U}{\mu}}\text{x } \frac{1}{\sqrt{x}} \text{ x } B \text{ x } dx = 0.365\mu U\sqrt{\frac{\rho U}{\mu}}\text{x } B\int_0^L x^{\frac{-1}{2}}\,\text{x}dx$$

$$= 0.365\mu U\sqrt{\frac{\rho U}{\mu}}\text{x } B\left[\frac{x^{1/2}}{1/2}\right]_0^L = 0.365\mu U\sqrt{\frac{\rho U}{\mu}}\text{x } B\sqrt{L}$$

$$F_D = 0.73\mu U B\sqrt{\frac{\rho U L}{\mu}}$$

$$\therefore C_D = \frac{0.73\mu U B\sqrt{\dfrac{\rho U L}{\mu}}}{\dfrac{1}{2}\rho A U^2}$$

(Where A = area of plate = L x B, L and B being width of the plate respectively)

$$\therefore C_D = \frac{0.73\mu UB\sqrt{\dfrac{\rho UL}{\mu}}}{\dfrac{1}{2}\rho \text{ x L x B x } U^2} = \frac{1.46\mu}{\rho LU}\sqrt{\frac{\rho UL}{\mu}}$$

$$\therefore \ \mathrm{C_D} = \frac{1.46\sqrt{\mu}}{\sqrt{\rho LU}} = 1.46\sqrt{\frac{\mu}{\rho UL}} = \frac{1.46}{\sqrt{\mathrm{Re}_L}}$$

Table 1-1 shows the values of δ (boundary layer thickness), C_D^* (local co-efficient of drag), C_D (average co-efficient of drag) in terms of Reynolds number (Re) for various velocity profiles/distributions.

Table1.1. Shows the values of δ, C_D^* and C_D in terms of Re

S. No.	*Velocity Profiles*	δ	C_D^*	C_D
1	$\frac{u}{U} = 2\left(\frac{y}{\delta}\right) - \left(\frac{y}{\delta}\right)^2$	$\frac{5.48x}{\sqrt{\mathrm{Re}_x}}$	$\frac{0.73}{\sqrt{\mathrm{Re}_x}}$	$\frac{1.46}{\sqrt{\mathrm{Re}_L}}$
2	$\frac{u}{U} = \frac{3}{2}\left(\frac{y}{\delta}\right) - \frac{1}{2}\left(\frac{y}{\delta}\right)^3$	$\frac{4.64x}{\sqrt{\mathrm{Re}_x}}$	$\frac{0.646}{\sqrt{\mathrm{Re}_x}}$	$\frac{1.292}{\sqrt{\mathrm{Re}_L}}$
3	$\frac{u}{U} = 2\left(\frac{y}{\delta}\right) - 2\left(\frac{y}{\delta}\right)^3 + \left(\frac{y}{\delta}\right)^4$	$\frac{5.84x}{\sqrt{\mathrm{Re}_x}}$	$\frac{0.686}{\sqrt{\mathrm{Re}_x}}$	$\frac{1.372}{\sqrt{\mathrm{Re}_L}}$
4	$\frac{u}{U} = \sin\left(\frac{\pi}{2}\frac{y}{\delta}\right)$	$\frac{4.795x}{\sqrt{\mathrm{Re}_x}}$	$\frac{0.654}{\sqrt{\mathrm{Re}_x}}$	$\frac{1.31x}{\sqrt{\mathrm{Re}_L}}$
5	*Blasius Results* $\mathrm{Re} \le 3.2x10^5$	$\frac{5x}{\sqrt{\mathrm{Re}_x}}$	$\frac{0.664}{\sqrt{\mathrm{Re}_x}}$	$\frac{1.328}{\sqrt{\mathrm{Re}_L}}$

Example1.10.

The boundary layer thickness at a distance of 1 m from the leading edge of a flat plate kept over zero angle of incidence to the flow direction is 1 mm. The velocity outside the boundary layer is 25 m/s. The boundary layer thickness at a distance of 4 m is (i) 4 mm, (ii) 2 mm, (iii) 1 mm.

Select the correct answer. Assume that the boundary layer is entirely laminar

Sol. Free stream velocity, U = 25 m/s

The boundary layer thickness at $X_1 = 1$ *m,* $\delta_1 = 1$ *mm*

The boundary layer thickness at a distance of 4 m, δ_2*:*

Thickness of boundary layer is given by,

$$\delta = \frac{5x}{\sqrt{\mathrm{Re}_x}} = \frac{5x}{\sqrt{\frac{Ux}{\nu}}} = 5\sqrt{\frac{\nu x}{U}}$$

$$= \delta_1 = 5\sqrt{\frac{\nu x_1}{U}}$$

$$= \delta_2 = 5\sqrt{\frac{\nu x_2}{U}}$$

$$\frac{\delta_1}{\delta_2} = \sqrt{\frac{x_1}{x_2}}$$

$$\frac{1}{\delta_2} = \sqrt{\frac{1}{4}} \qquad \text{or} \quad \delta_2 = 2\,\text{mm} \qquad \text{(Ans.)}$$

Example 13-15.

Air is flowing over a smooth flat plate with a velocity of 12 m/s. The velocity profile is in the form: $\frac{u}{U} = 2\left(\frac{y}{\delta}\right) - \left(\frac{y}{\delta}\right)^2$

The length of the plate is 1.1 m and width 0-9 m. If laminar boundary layer exists up to a value of Re =2 x l0^5 *and kinematic viscosity of air is 0.15 stokes, find:*

(i) *The maximum distance from the leading edge upto which laminar boundary layer exists, and*

(ii) *The maximum thickness of boundary layer.*

Solution:

Velocity profile (distribution): $\frac{u}{U} = 2\left(\frac{y}{\delta}\right) - \left(\frac{y}{\delta}\right)^2$

Velocity of air, U=12m/s

Length of plate, L=1.1m

Width of the plate, B=0.9

Reinhold's number upto which laminar boundary exists, Re = 2x10^5

Kinematic viscosity of air, V=0.15 stokes = 0.15 x 10^{-4}*m*^{2}*/s*

(i) *the maximum distance from the leading edge upto which laminar boundary layer exists, x:*

$$\mathrm{Re}_x = \frac{Ux}{\nu} \qquad \text{or} \quad 2\text{x}10^5 = \frac{12\text{ x }x}{0.15\text{ x }10^{-4}}$$

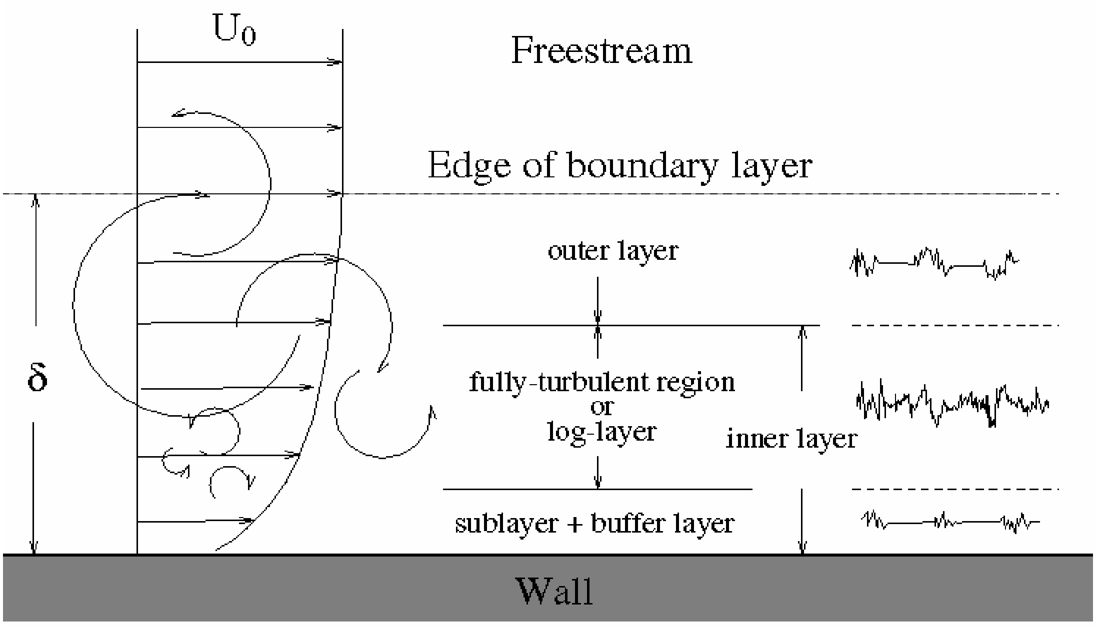
$$x = \frac{2\text{x}10^5\text{ x}0.15\text{ x }10^{-4}}{12} = 0.25m$$

(ii) *the maximum thickness of the boundary layer, δ:*

for the given velocity profile, the maximum thickness of the boundary layer is given by, $\delta = \frac{5.48x}{\sqrt{\mathrm{Re}_x}}$

$$\delta = \frac{5.48\text{x}0.25}{\sqrt{2\text{x}10^5}} = 0.00306m \quad \text{or} \quad 3.06\text{mm} \quad \text{(Ans.)}$$

13-5. Turbulent Boundary Layer:



As compared to laminar boundary layers, the turbulent boundary layers are thicker. Further in a turbulent boundary layer the velocity distribution is much more uniform, than in a laminar boundary layer, due to intermingling of fluid particles between different layers of the fluid. The velocity distribution in a turbulent boundary layer follows a logarithmic law i.e. u ~ log y, which can also be represented by a power law of the type:

$\frac{u}{U}=\left(\frac{y}{\delta}\right)^{n}$*, where* $n=\frac{1}{7}$ *(approx.) for Re<*10^7 *but >5x*10^5

$$\therefore \frac{u}{U}=\left(\frac{y}{\delta}\right)^{\frac{1}{7}}$$

This is known as one-seventh power law.

Eqn. (1-40), however, cannot be applied at the boundary itself because at $y=0,\ \left(\frac{\delta u}{\delta y}\right)=\frac{1}{7}U\delta^{\frac{1}{7}}y^{\frac{-6}{7}}=\infty$. *This difficulty is circumvented by considering the velocity in the viscous laminar sublayer to be linear and tangential to the seventh-root*

profile at the point, where the laminar sub layer merges with the turbulent part of the boundary layer.

Blasius suggested the following relation for viscous shear stress:

$$\tau_o = 0.0226 \rho U^2 \left(\frac{\mu}{\rho U \delta} \right)^{\frac{1}{4}}$$

(for Re ranging from 5 x 10^5 to 10^7)

Let us now find the values of δ, τ_o, C^*, F_D *and* C_d *for the velocity distribution given by eqn. (13-40) i.e.* $\left[\frac{u}{U} = \left(\frac{y}{\delta} \right)^{\frac{1}{7}} \right]$.

Boundary layer thickness, δ:

Substituting the value of $\frac{u}{U}$ *in Von Karman integral eqn. (13-6), we have*

$$\frac{\tau_o}{\rho U^2} = \frac{d}{dx} \left[\int_0^{\delta} \frac{u}{U} \left(1 - \frac{u}{U} \right) dy \right]$$

$$\frac{\tau_o}{\rho U^2} = \frac{d}{dx} \left[\int_0^{\delta} \left(\frac{y}{\delta} \right)^{\frac{1}{7}} \left(1 - \left(\frac{y}{\delta} \right)^{\frac{1}{7}} \right) dy \right]$$

$$\frac{\tau_o}{\rho U^2} = \frac{d}{dx} \left[\int_0^{\delta} \left(\frac{y}{\delta} \right)^{\frac{1}{7}} - \left(\frac{y}{\delta} \right)^{\frac{2}{7}} dy \right]$$

$$\frac{\tau_o}{\rho U^2} = \frac{d}{dx} \left[\frac{7}{8} \frac{y^{\frac{8}{7}}}{\delta^{\frac{1}{7}}} - \frac{7}{9} \frac{y^{\frac{9}{7}}}{\delta^{\frac{2}{7}}} \right]_0^{\delta} = \frac{d}{dx} \left[\frac{7\delta}{8} - \frac{7\delta}{9} \right] = \frac{7}{72} \frac{d\delta}{dx}$$

[In the expression above, the limits have been taken from 0 to δ instead of δ' to δ since the laminar sub layer (δ') is very thin]

$$\tau_o = \frac{7}{72}\rho U^2 \frac{d\delta}{dx}$$

Also, $$\tau_o = 0.0225\rho U^2 \left(\frac{\mu}{\rho U\delta}\right)^{\frac{1}{4}}$$

Now equating the eqns. (13-42) and (13-41), we have

$$\tau_o = \frac{7}{72}\rho U^2 \frac{d\delta}{dx} = 0.0225\rho U^2 \left(\frac{\mu}{\rho U\delta}\right)^{\frac{1}{4}}$$

Or $$\frac{7}{72}\frac{d\delta}{dx} = 0.0225\left(\frac{\mu}{\rho U}\right)^{\frac{1}{4}} x \frac{1}{\delta^{\frac{1}{4}}}$$ *, (canceling ρU^2 on both sides)*

$$\delta^{\frac{1}{4}} d\delta = 0.0225x\frac{72}{7}x\left(\frac{\mu}{\rho U}\right)^{\frac{1}{4}} dx$$

$$\delta^{\frac{1}{4}} d\delta = 0.2314\left(\frac{\mu}{\rho U}\right)^{\frac{1}{4}} dx$$

Integrating both sides, we have

$$\frac{4}{5}\delta^{\frac{5}{4}} = 0.2314\left(\frac{\mu}{\rho U}\right)^{\frac{1}{4}} x + C$$ *(Where C = constant of integration)*

Let boundary layer be assumed to be turbulent over the entire length of plate.

Hence, at *x=0, δ=0* $\therefore C = 0$

$$\therefore \frac{4}{5}\delta^{\frac{5}{4}} = 0.2314\left(\frac{\mu}{\rho U}\right)^{\frac{1}{4}} x$$ *Or* $$\delta^{\frac{5}{4}} = (5/4\text{x}0.2314)\left(\frac{\mu}{\rho U}\right)^{\frac{1}{4}} \text{x}\ x$$

$$\delta = [5/4\text{x}0.2314]^{\frac{4}{5}}\left(\frac{\mu}{\rho U}\right)^{\frac{1}{5}} \text{x}\ x^{\frac{4}{5}}$$

$$\delta = 0.371\left(\frac{\mu}{\rho U x}\right)^{\frac{1}{5}} x^{\frac{1}{5}} \text{ x } x^{\frac{4}{5}} = 0.0371\left(\frac{1}{\text{Re}_x}\right)^{\frac{1}{5}} \text{ x } x = \frac{0.371x}{(\text{Re}_x)^{\frac{1}{5}}}$$

$$\delta = \frac{0.371x}{(\text{Re}_x)^{\frac{1}{5}}}$$

(iii) shear stress, τ_o:

$$\tau_o = 0.0225\rho U^2\left(\frac{\mu}{\rho U \delta}\right)^{\frac{1}{4}}$$

Substituting the value of δ from Eqn. (13.43), we get

$$\tau_o = 0.0225\rho U^2\left(\frac{\mu}{\rho U \dfrac{0.371x}{\text{Re}_x^{\frac{1}{5}}}}\right)^{\frac{1}{4}}$$

$$\tau_o = \frac{0.0225}{0.371^{\frac{1}{4}}}\rho U^2\left[\frac{\mu}{\rho U x}(\text{Re}_x)^{\frac{1}{5}}\right]^{\frac{1}{4}} = 0.0288\rho U^2\left[\frac{(\text{Re}_x)^{\frac{1}{5}}}{\text{Re}_x}\right]^{\frac{1}{4}},$$

$$\because \frac{1}{\text{Re}_x} = \frac{\mu}{\rho U x}$$

Or $$\tau_o = \frac{\rho U^2}{2} \text{ x } \frac{0.0576}{(\text{Re}_x)^{\frac{1}{5}}}$$

(iii) Local co-efficient of drag, C_D^*:

We know $$\tau_o = \frac{\rho U^2}{2} \text{ x } \frac{0.0576}{(\text{Re}_x)^{\frac{1}{5}}}$$

Also $\tau_o = C_D^* \text{ x } \frac{1}{2}\rho U^2$

Now equating the eqns. (13.44) and (13.9), we have

$$C_D^* \text{ x } \frac{1}{2}\rho U^2 = \frac{\rho U^2}{2} \text{ x } \frac{0.0576}{(\text{Re}_x)^{\frac{1}{5}}} \quad or \quad C_D^* = \frac{0.0576}{(\text{Re}_x)^{\frac{1}{5}}}$$

(iv) Drag force, F_D*:*

The total drag force (F_D*) on one side of the plate of width B and length L is given by,*

$$F_D = \int_0^L \tau_o \text{ x B x } dx$$

$$F_D = \int_0^L \frac{\rho U^2}{2} \text{ x } \frac{0.0576}{(\text{Re}_x)^{\frac{1}{5}}} \text{x } B \text{ x } dx = \int_0^L \frac{\rho U^2}{2} \text{ x } \frac{0.0576}{\left(\frac{\rho U x}{\mu}\right)^{\frac{1}{5}}} \text{x } B \text{ x } dx$$

$$= \frac{\rho U^2}{2} \text{ x } \frac{0.0576}{\left(\frac{\rho U}{\mu}\right)^{\frac{1}{5}}} B\int_0^L x^{\frac{-1}{5}} dx = \frac{\rho U^2}{2} \text{ x } \frac{0.0576}{\left(\frac{\rho \text{U}}{\mu}\right)^{\frac{1}{5}}} B\left[\frac{5}{4} \text{ x } x^{\frac{4}{5}}\right]_0^L$$

$$= \frac{\rho U^2}{2} \text{ x } \frac{0.0576}{\left(\frac{\rho U}{\mu}\right)^{\frac{1}{5}}} \text{ x } B \text{ x } \frac{5}{4}(L)^{\frac{4}{5}} = \frac{\rho U^2}{2} \text{ x } \frac{0.072}{\left(\frac{\rho UL}{\mu}\right)^{\frac{1}{5}}} \text{ x } B \text{ x L}$$

Or $$F_D = \frac{\rho U^2}{2} \text{ x } \frac{0.072}{(\text{Re}_L)^{\frac{1}{5}}} \text{ x } B \text{ x } L$$

(v)Co-efficient of drag, C_D*:*

We know, $F_D = C_D \text{ x } \frac{1}{2} \rho A V^2$

Now equating eqns. (13.10) and (13.46), we have

$$C_D \text{ x } \frac{1}{2} \rho \text{ x } B \text{ x L x U}^2 = \frac{\rho U^2}{2} \cdot \frac{0.072}{(\text{Re}_L)^{1/5}} \text{ x } B \text{ x L}$$

Or $$C_D = \frac{0.072}{(\text{Re}_L)^{1/5}}$$

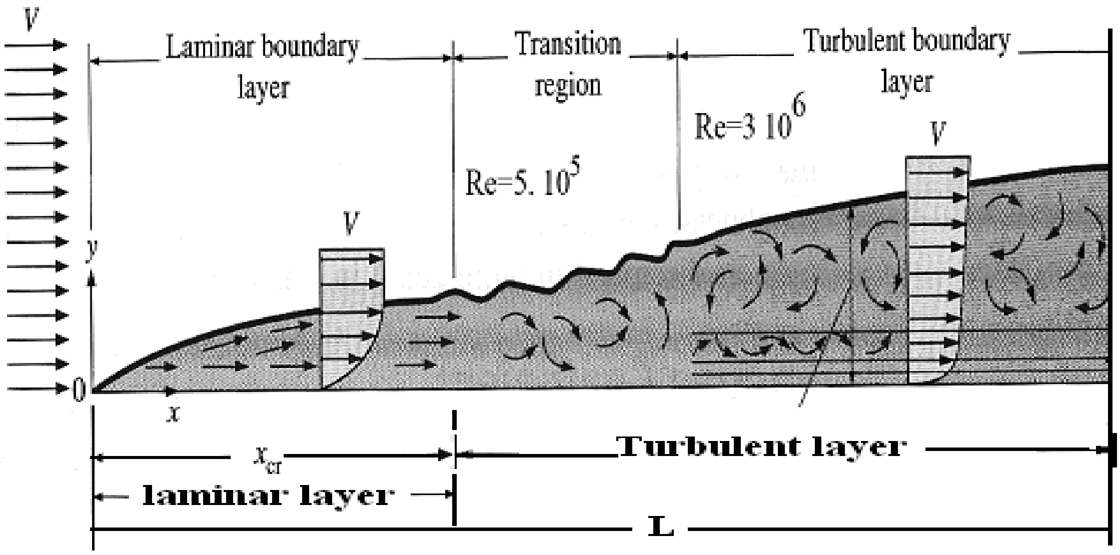
This is valid for $5 \text{ x } 10^5 \leq \text{Re}_L \leq 10^7$

For Reynolds number between 10^7 *and* 10^9 *the following relationship suggested by Prandtl and Schlichting holds good.*

$$C_D = \frac{0.455}{(\log_{10} \text{Re}_L)^{2.58}}$$

13-6. Total Drag due to Laminar and Turbulent Layers

When the leading edge is not very rough, the turbulent boundary layer does not begin at the leading edge, it is usually preceded by the laminar boundary layer. The point of transition from laminar to turbulent layer depends upon the intensity of turbulence. The distance x_c *(Fig. 13-7) of the transition from the leading edge can be obtained from critical Reynolds number which normally ranges from* $3 \text{ x } 10^5$ *to* $3 \text{ x } 10^6$*.*



13.7. Drag due to laminar and turbulent boundary layers.

Drag force (F_D = F) for the turbulent boundary layer can be estimated from the following relation:

$$F_{turb} = (F_{turb.})_{total} - (F_{turb.})_{x_c}$$

Where **(F_{uture})$_{total}$** *= the drag which would occur if a turbulent boundary extends along the entire length of the plate, and*

(F_{trub})x_c *= the drag due to fictitious turbulent boundary layer from the leading edge to a distance* $\boldsymbol{x_c}$*.*

Let us assume that the plate is long enough so that Reynolds number is greater than 10^7, then the turbulent drag is given by,

$$F_{turb.} = \frac{0.455}{(\log_{10} \mathrm{Re}_L)^{2.58}} \text{ x } \frac{\rho U^2}{2} \text{ x } L \text{ x B} - \frac{0.072}{(\mathrm{Re}_c)^{\frac{1}{5}}} \text{ x } \frac{\rho \mathrm{U}^2}{2} \text{ x } x_c \text{ x B}$$

Where:

L = length of the plate,

B = width of the plate, and

U = free stream velocity.

The laminar boundary layer prevails within the length x_c and its contribution to drag force is given by

$$F_{laminar} = \frac{1.328}{\sqrt{\mathrm{Re}_c}} \text{ x } \frac{\rho \mathrm{U}^2}{2} \text{ x } x_c \text{ x B} = \frac{1.328 x_c}{\sqrt{\mathrm{Re}_c}} \text{ x B x } \frac{\rho \mathrm{U}^2}{2}$$

$F_{total} = F_{laminar} + F_{turbulent}$

$$F_{total} = \frac{1.328 x_c}{\sqrt{\mathrm{Re}_c}} \text{ x B x } \frac{\rho \mathrm{U}^2}{2} + \left[\frac{0.455 L}{(\log_{10} \mathrm{Re}_L)^{2.58}} \text{ x B x } \frac{\rho \mathrm{U}^2}{2} - \frac{0.072 x_c}{(\mathrm{Re}_c)^{1/5}} \text{ x B x } \frac{\rho \mathrm{U}^2}{2} \right]$$

$$F_{total} = \left[\frac{1.328 x_c}{\sqrt{\mathrm{Re}_c}} + \frac{0.455 L}{(\log_{10} \mathrm{Re}_L)^{2.58}} - \frac{0.072 x_c}{(\mathrm{Re}_c)^{1/5}} \right] \frac{B \rho U^2}{2}$$

Also, $\dfrac{\mathrm{Re}_c}{\mathrm{Re}_L} = \dfrac{(\rho U x_c / \mu)}{(\rho U L / \mu)} = \dfrac{x_c}{L}$ *Or* $x_c = \dfrac{\mathrm{Re}_c\, L}{\mathrm{Re}_L}$

Substituting the value of $\mathbf{x_c}$ *in eqn. (iii), we have*

$$F_{total} = \left[\frac{1.328\sqrt{\mathrm{Re}_c}}{\mathrm{Re}_L} + \frac{0.455}{(\log_{10} \mathrm{Re}_L)^{2.58}} - \frac{0.072\,\mathrm{Re}_c^{0.8}}{(\mathrm{Re}_c)^{1/5}} \right] \frac{LB\rho U^2}{2}$$

Assuming that transition occurs at $Re_c = 5x10^5$

$$\therefore \quad F_{total} = C_D \mathrm{x} \frac{1}{2} \rho A U^2 = C_D \mathrm{x} \frac{\mathrm{LB}\rho \mathrm{U}^2}{2}$$

(where C_D *= average co-efficient of drag.), Equating the above two equations*

$$C_D = \frac{0.455}{(\log_{10} \mathrm{Re}_L)^{2.58}} - \frac{1700}{\mathrm{Re}_L}$$

Example 13-24

A submarine can be assumed to have cylindrical shape with rounded nose. Assuming its length to be 50 m and diameter 5.0 m, determine the total power required to overcome boundary friction if it cruises at 8 m/s velocity in sea water at 20°C (ρ = 1030 kg/m^3*), ν=1x10*$^{-6}$*m*^{2}*/s.*

Sol. *Length of submarine, L = 50 m, Diameter of submarine, D = 50 m*

Velocity of submarine, U = 8 m/s, Density of sea water, ρ= 1030 kg/m^3

Kinematic viscosity of sea water, ν= 1 x 10^{-6} *m*^{2}*/s*

Total power required to overcome boundary friction, P:

Reynolds number, $\mathrm{Re}_\mathrm{L} = \dfrac{UL}{\nu} = \dfrac{8\mathrm{x}50}{1\mathrm{x}10^{-6}} = 4\mathrm{x}10^8$

The length over which boundary layer will be laminar is given by,

$$\frac{Ux}{\nu} = 5\mathrm{x}10^5 \quad \text{or} \quad x = \frac{5\mathrm{x}10^5\,\mathrm{x}\,\nu}{\mathrm{U}}$$

$$\therefore \qquad x = \frac{5x10^5 \, x1x10^{-6}}{8} = 0.0625m$$

This being very small, contribution to total drag from laminar boundary layer is negligible; hence C_D is given by,

$$C_D = \frac{0.455}{(\log_{10} \mathrm{Re}_L)^{2.58}} = \frac{0.455}{\left[\log_{10}\left(4x10^8\right)\right]^{2.58}} = 0.001765$$

Area, $\qquad A = \pi DL = \pi \text{ x } 5\text{x } 50 = 785.4m^2$

$$\therefore \text{Drag Force,} \qquad \mathrm{F_D} = C_D \text{ x } \frac{1}{2}\rho A U^2 = 0.001765\text{x}\frac{1}{2}x1030\text{x}785.4\text{x}8^2 = 456902N$$

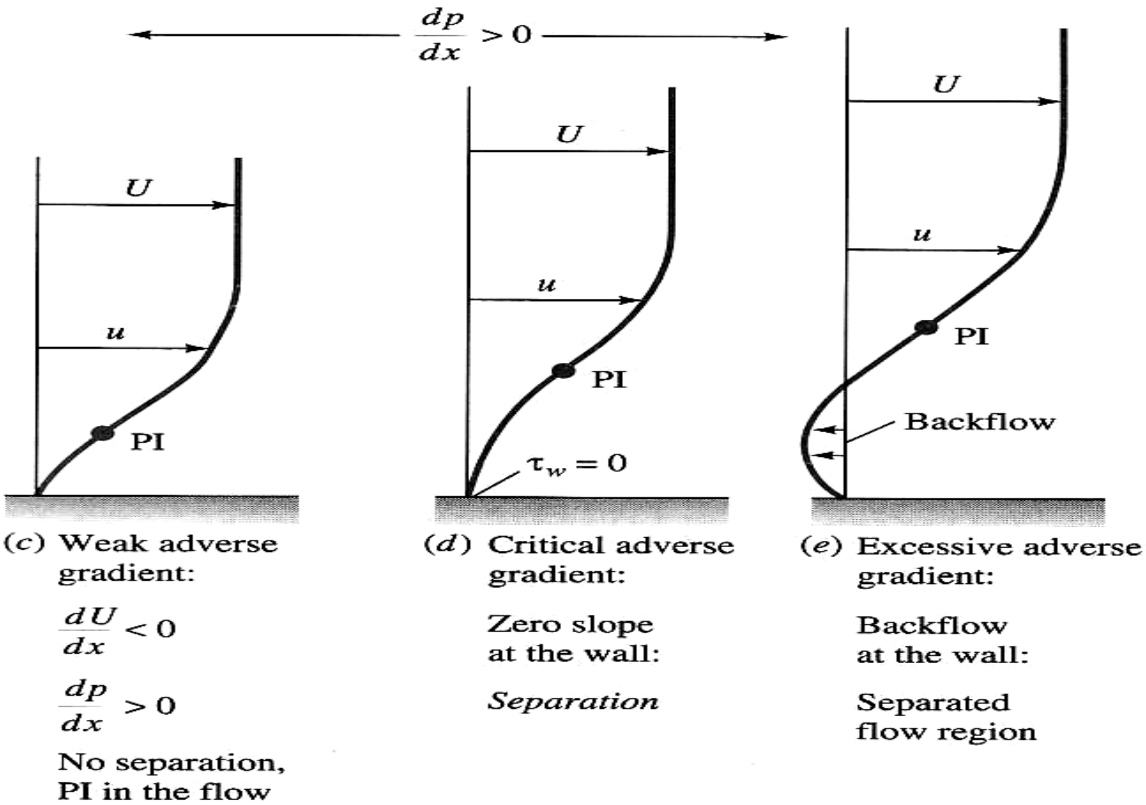
Hence total power required to overcome boundary friction,

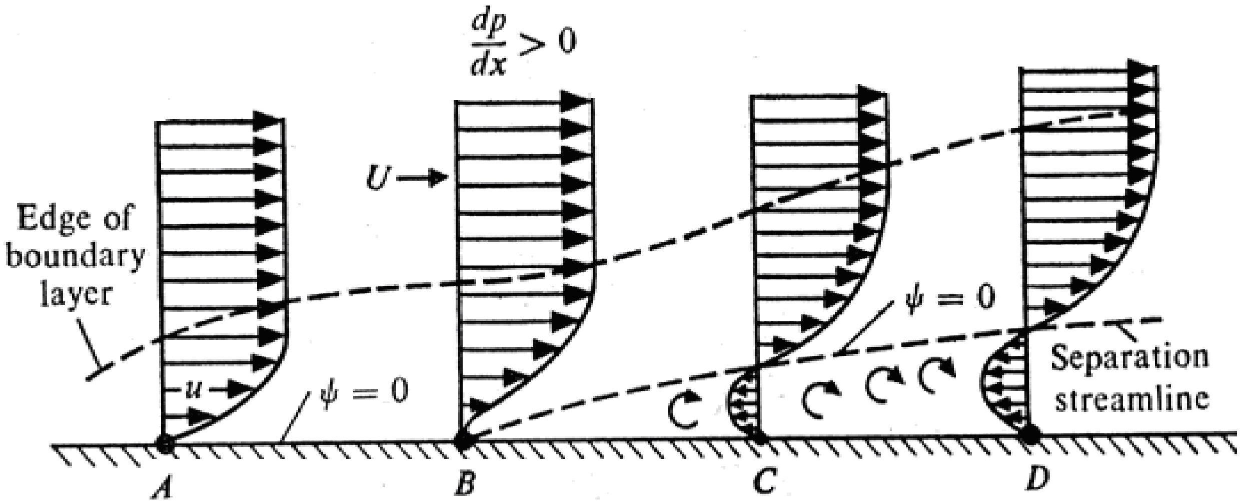
$$P = \frac{F_D U}{1000} kW = \frac{45690.2\text{x}8}{1000} = 365.52kW \text{ (Ans)}$$

13.7 Separation

What causes separation?

The increasing downstream pressure slows down the wall flow and can make it go backward-flow separation. $dp/dx > 0$, *adverse pressure gradient, flow separation may occur.* $dp/dx < 0$, *favorable gradient, flow is very resistant to separation. Previous analysis of BL was valid before separation.*





Note: 1. Due to backflow close to the wall, a strong thickening of the BL takes place and BL mass is transported away into the outer flow

2. At the point of separation, the streamlines leave the wall at a certain angle.

Boundary Layer Separation and its Control

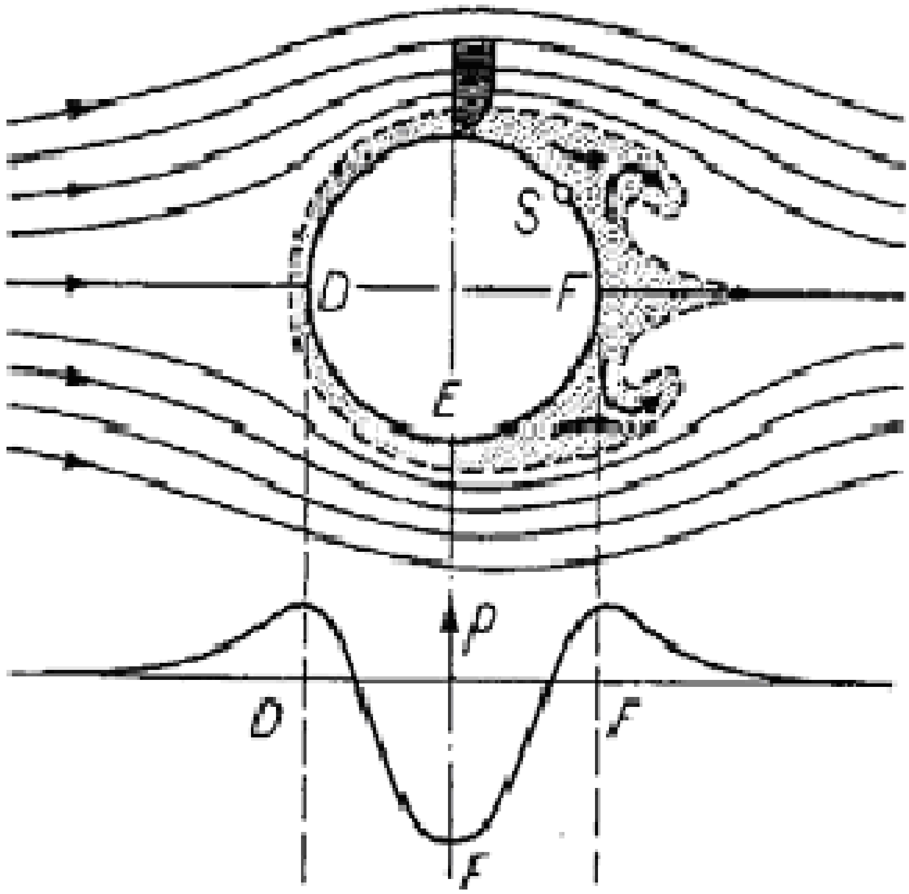


Fig. 1.8 Separation of the boundary layer and vortex formation at a circular cylinder (schematic). S = separation point

Notes:

1. D to E, pressure drop, pressure is transformed into kinetic energy.

2. From E to F, kinetic energy is transformed into pressure.

3. A fluid particle directly at the wall in the boundary layer is also acted upon by the same pressure distribution as in the outer flow (inviscid).

4. Due to the strong friction forces in the BL, a BL particle loses so much of its kinetic energy that is cannot manage to get over the “pressure gradient” from E to F.

5. The following figure shows the time sequence of this process:

a. reversed motion begun at the trailing edge

b. boundary layer has been thickened, and start of the reversed motion has moved forward considerably.

c. and d. a large vortex formed from the backflow and then soon separates from the body.

In a flowing fluid when a solid body is immersed, a thin layer of fluid called the boundary layer is formed adjacent to the solid body. The forces acting on the fluid in the boundary layer are:

1- Inertia forces, 2- Viscous forces and 3- Pressure force.

— *When the pressure gradient in the direction of flow is negative* $\left[\frac{dp}{dx}\langle 0\right]$ *i.e. when the pressure decreases in the direction of flow, the flow is accelerated. In this case, the pressure force and inertia force add together and jointly tend to reduce the effect of viscous forces in the boundary layer. This results in a decrease in the thickness of boundary layer in the direction of flow, as a consequence of which there are low losses and high efficiencies in accelerating flows.*

— *When the pressure increases in the direction of flow* $\left[\frac{dp}{dx}\rangle 0\right]$ *the pressure forces acts opposite to the direction of flow and further increases the retarding effect of the viscous forces. Subsequently the thickness of the boundary layer increases rapidly in the direction of flow. If these forces act over a long stretch, the boundary layer gets separated from the surface and moves into the main stream. This phenomenon is called* ***separation***. *The point of the body at which the boundary layer is on the verge of separation from the surface is called point of separation.*

Consider a flow of fluid over a curved surface as shown in Fig.1.8.

— *As the fluid flows round the surface (the area of flow decreases) it is accelerated over the left hand section until at point B the velocity just outside the boundary is maximum and the pressure is minimum (as shown by the graph below the surface).*

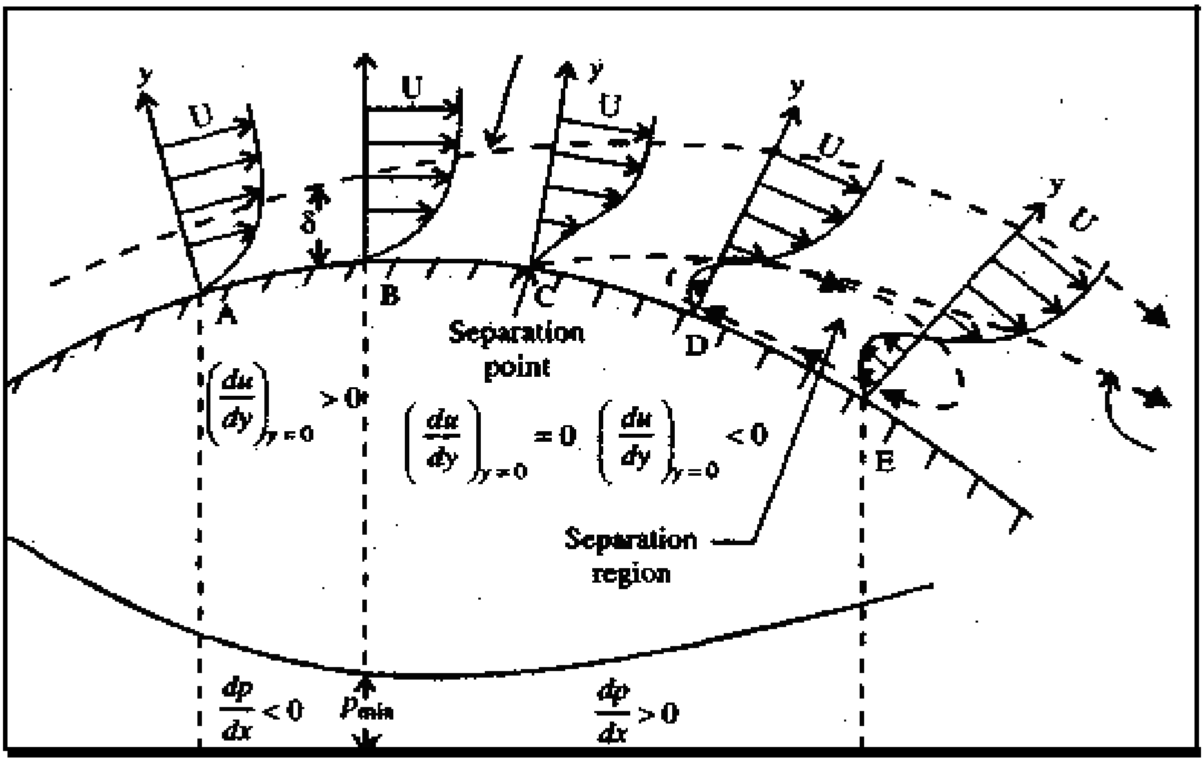


Fig.13.8. Separation of boundary layer.

Edge of the y boundary layer Separations stream line A and B the pressure gradient is negative, As long as $\left[\frac{dp}{dx}\langle 0\right]$*, the entire boundary layer moves forward. Beyond B (i.e. along the region BCDE), the area of flow increases and hence velocity of flow decreases; due to decrease of velocity the pressure increases (in the direction of flow) and hence the pressure gradient* $\left[\frac{dp}{dx}\right]$ *is positive i.e.* $\left[\frac{dp}{dx}\rangle 0\right]$*. The value of the velocity gradient* $\left[\frac{du}{dy}\right]$ *at the boundary is zero at the point C, this point is known as a* ***separation point*** *(the boundary layer starts separating from the surface because further retardation of flow near the surface is physically impossible). Large turbulent eddies are formed downstream of the point of separation. The disturbed region in which the eddies are formed is called* ***turbulent wake****.*

The flow separation depends upon factors such as:

1- The curvature of the surface;

2- The Reynolds number of flow;

3- The roughness of the surface.

The velocity gradient, for a given velocity profile, exhibits the following characteristics for the flow to remain attached, get detached or be on the verge of separation.

1- $\left(\frac{\partial u}{\partial y}\right)_{y=0}$ is positive ..... Attached flow (the flow will not separate)

2- $\left(\frac{\partial u}{\partial y}\right)_{y=0}$ is zero ..... The flow is on the verge of separation

3- $\left(\frac{\partial u}{\partial y}\right)_{y=0}$ is negative ..... separated flow.

- Boundary layer separation is unstable, inefficient process and entails large losses due to appreciable eddying region.

-Separation occurs in the following cases:

1. Diffusers,

2. Open channel transitions,

3. Pumps,

4. Fans,

5. Aerofoil,

6. Turbine blades etc.

Examples of BL Separations (two-dimensional):

***Features**: The entire boundary layer flow breaks away at the point of zero wall shear stress and, having no way to diverge left or right, has to go up and over the resulting separation bubble or wake.*

1. Plane wall(s)

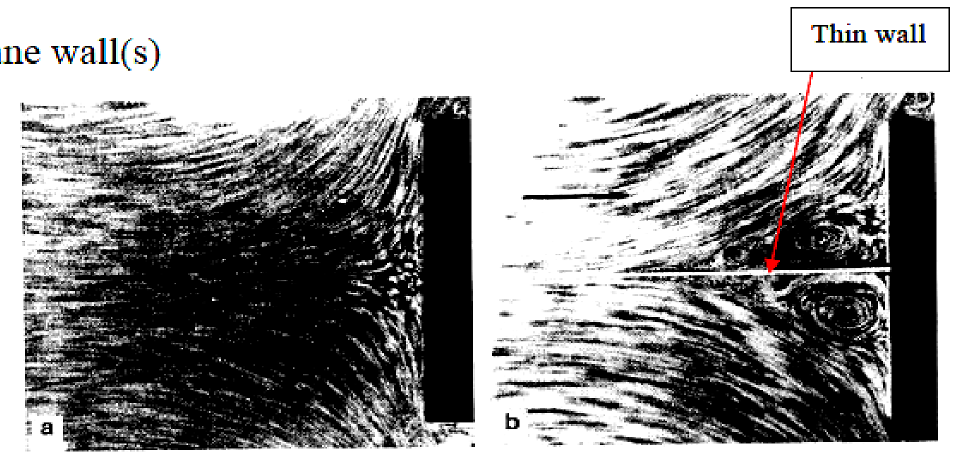
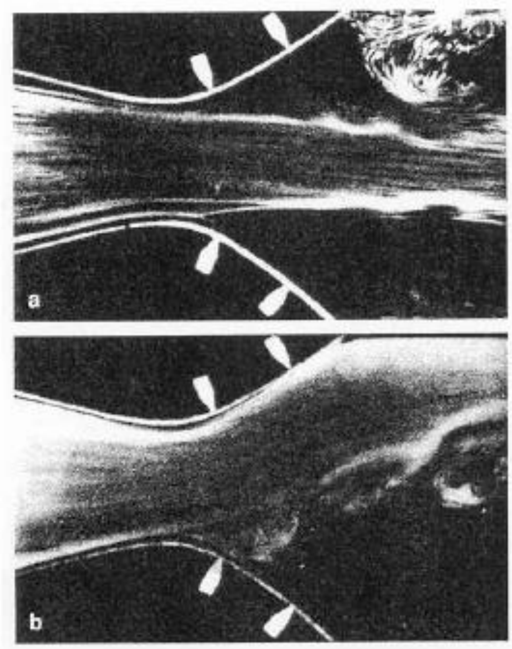


Fig. 2.10. Stagnation point flow, after H. Föttinger (1939), (a) free stagnation–point flow without separation, (b) retarded stagnation–point flow, with separation

(a). Plane stagnation-point flow: no separation on the streamlines of symmetry (no wall friction present), and no separation at the wall (favorable pressure gradient)

(b).Flat wall with right angle to the wall: flow separate, why?

2. Diffuser flow:



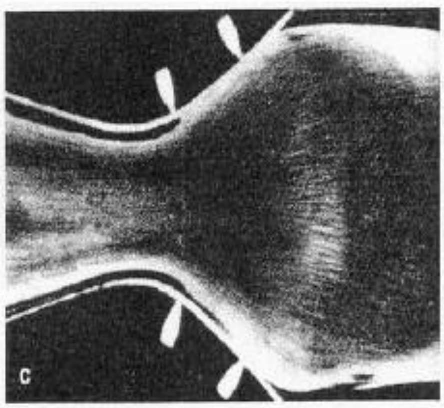


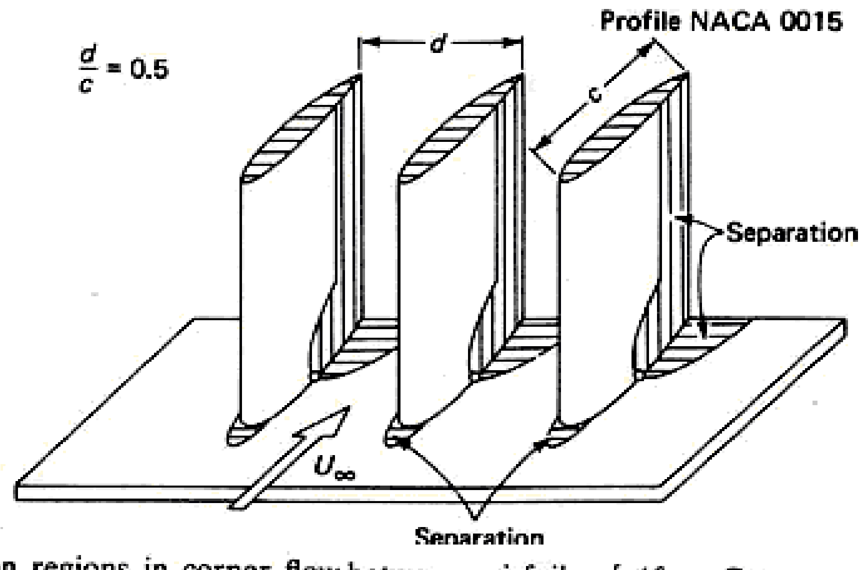
Fig. 2.9. Flow in a widening channel (diffuser) (a) separation at both diffuser walls, (b) suction of the boundary layer at the upper diffuser wall, (c) suction at both diffuser walls (after L. Prandtl; O. Tietjens (1931))


(a)


(b)

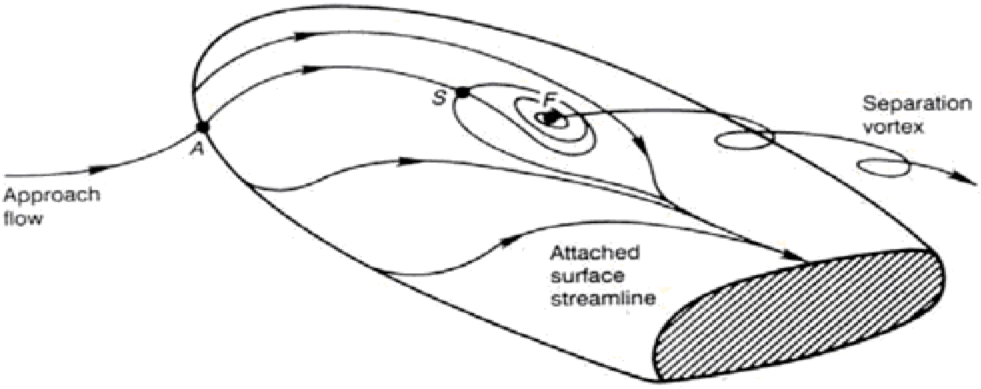
3. Turbulent Boundary Layer

Influence of a strong pressure gradient on a turbulent flow
(a) a strong negative pressure gradient may re-laminarize a flow
(b) a strong positive pressure gradient causes a strong boundary layer top thicken.
Examples of BL Separations (three-dimensional)
Features: unlike 2D separations, 3D separations allow many more options.
There are four different special points in separation:
(1). A nodal Point, where an infinite number of surfaces streamline merged tangentially to the separation line.
(2). A saddle point, where only two surface streamlines intersect and all others divert to either side
(3). A focus, or spiral node, which forms near a saddle point and around which an infinite number of surface streamlines swirl.
(4). A three-dimensional singular point, not on the wall, generally serving as the center for a horseshoe vortex.



Separation regions in corner flow between airfoils. [*After* *Gersten* (*1959*).]

1. ***3D separations on a round-nosed body at angle of attack***



Three-dimensional separation on a round-nosed body at angle of attack, first described by Legendre (1965). Point A is a nodal attachment point, point S is a saddle point, and point F is a focus of separation.

Methods of preventing the separation of boundary Layer:

Following are some of the methods generally adopted to retard or arrest the flow separation:

1. *Streamlining the body shape.*
2. *Tripping the boundary layer from laminar to turbulent by provision of surface roughness.*
3. *Sucking the retarded flow.*
4. *Injecting high velocity fluid in the boundary layer.*
5. *Providing slots near the leading edge.*
6. *Guidance of flow in a confined passage.*
7. *Providing a rotating cylinder near the leading edge.*
8. *Energizing the flow by introducing optimum amount of swirl in the incoming flow. (Note: Refer Example 13.29 also)*

Example.13.29

Explain what is meant by separation of boundary layer. Describe with sketches the methods to control separation.

Sol.

When a solid body is immersed in a flowing fluid, a thin layer of fluid called boundary layer is formed, adjacent to the solid body. In this thin layer of fluid, the

velocity varies from zero to free stream velocity in the direction normal to the solid body. Along the length of the solid body, the thickness of the boundary layer increases. The fluid layer adjacent to the solid surface has to do work against surface friction at the expense of kinetic energy. This loss of kinetic energy is recovered from the immediate fluid layer in contact with the layer adjacent to the solid surface through momentum exchange process. Thus the velocity of the layer goes on decreasing. Along the length of solid body, at a certain point a stage may come when the boundary layer may not be able to keep sticking to the solid body, if it cannot provide kinetic energy to overcome the resistance offered by the solid body. In other words, the boundary layer will be separated from the surface. This phenomenon is called the ***boundary layer separation****. The point on the body at which the boundary layer is on the verge of separation from the surface is called* ***point of separation****.*

Methods to control separation:

1. Motion of solid boundary:

By rotating a circular cylinder lying in a stream of fluid, so that the upper side of cylinder where the fluid as well as the cylinder moves in the same direction, the boundary layer does not form. However on the lower side of cylinder where the fluid motion is opposite to that of cylinder separation would occur (Fig. 13.9).

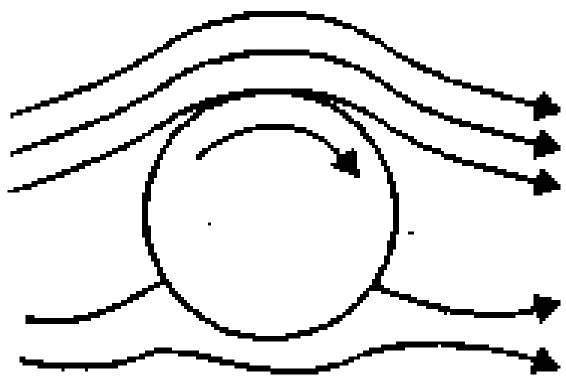


Fig. 13.9

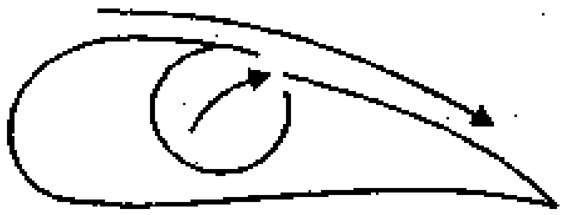


Fig.13.10. Injection fluid into boundary layer

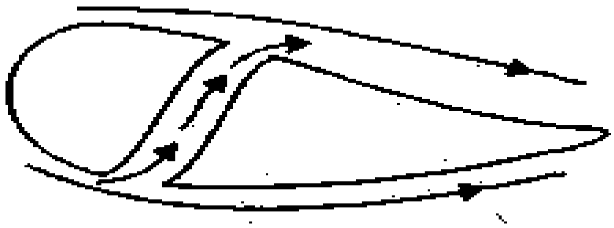


Fig.3.11 slotting wing

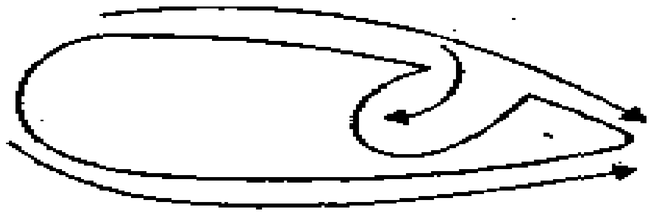


Fig.13.12 suction of fluid from boundary layer

2. Acceleration of fluid in the boundary layer:

This method of controlling separation consists of supplying additional energy to particles of fluid which are being retarded in the boundary layer. This may be achieved either by injecting the fluid into the region of boundary layer from the interior of the body with the help of some available device as shown in Fig. 13.10 or by diverting a portion of fluid of the main stream from the region of high pressure to the retarded region of boundary layer through a slot provided in the body (fig. 13.11)

3. Suction of fluid from the boundary layer:

In this method, the slow moving fluid in the boundary layer is removed by suction through slots or through a porous surface as shown in the Fig. 13.12.

4. Streamlining of body shapes:

By the use of suitably shaped bodies the point of transition of the boundary layer from laminar to turbulent can be moved downstream which results in the reduction of

the skin friction drag. Furthermore by streamlining of body shapes, the separation may be eliminated.

1.1.5. Comments on "Boundary layer, pressure gradient, and separation"

- Fluid in the BL in the region near wall is continuously subjected to (net) shear stress (acting on fluid in the –x direction) from the start of the leading edge. (See Fig. 2a.)

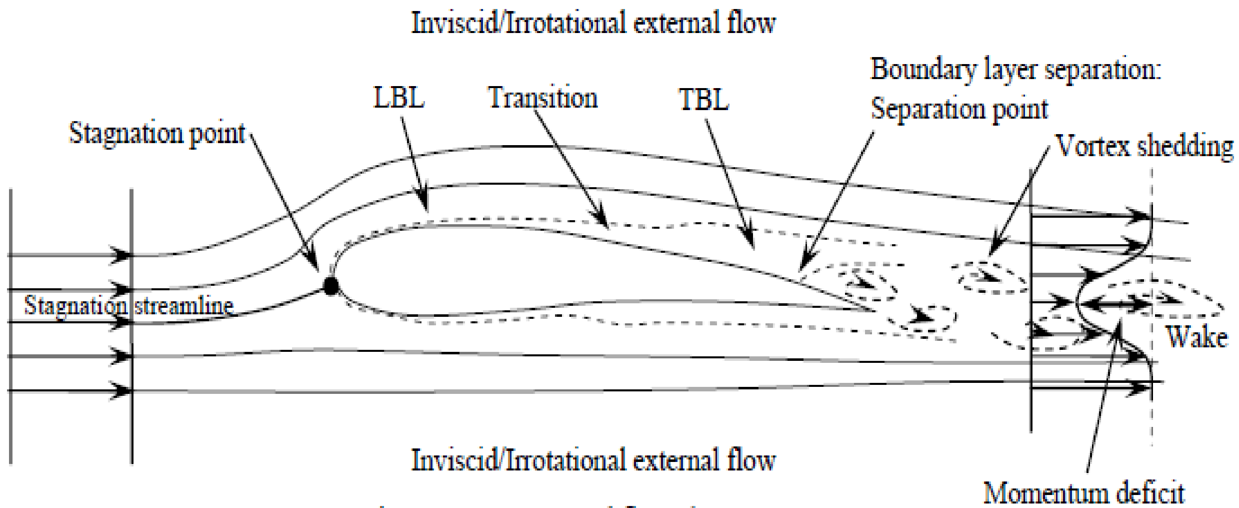


Fig. 1. Some external flow phenomena.

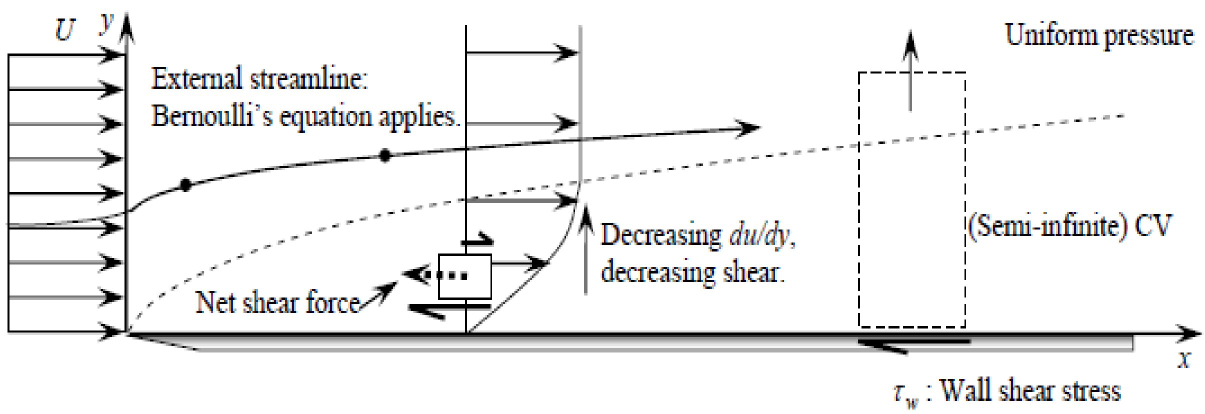


Fig. 2. Boundary layer as a layer of viscous/shear/rotational flow. Two views of shear: fluid element and CV. External flow is often inviscid and irrotational (if in fact the fluid is static and the body is moving).

1.1.6. Wake: Region of high shear/friction behind a body. Vortex shedding.

- *Fluid that was subjected to shear/viscous force in the BL is convected downstream from the body, forming another viscous/shear layer called wake.*

- *Bulk of rotational fluids from the upper and lower BL's often leave the upper and lower surfaces alternately, resulting in a phenomenon called vortex shedding.*
- *Vortex shedding: shedding of BL's vortices.*
- *Similar to BL, one can think of mass and momentum deficits in the wake.*

1.1.7. Aerodynamics: Aerodynamic forces and moments (lift, drag, etc.):

- *They are simply resultants of stress distribution (pressure + friction) due to fluid on solid surface. So long as there is no large-scale separation:*
- *Lift is due mainly to pressure and can be estimated by inviscid effect/flow.*
- *Drag is due mainly to wall shear stress (skin friction drag) and is the result of viscous effect. If there is large-scale separation:*
- *Loss of lift.*
- *Pressure distribution is such that it causes relatively larger amount of drag (pressure drag) than wall shear stress (skin friction drag).*
- *Total drag D = Skin friction drag (D_f, τ_w) + Pressure drag (D_p).*
- *Streamline Body and Bluff Body (and The In-Between): [Phenomenon wise, whether a body is streamline or bluff depends upon how it appears to the free stream, not to us. Hence, a well-designed, streamlined airfoil is a bluff body if it moves in a free stream upright or at very high angle of attack.]*
- *Streamline body: No large-scale separation:*
- *Drag of streamline body is due mainly to wall shear stress (skin friction drag).*
- *Bluff body: Large-scale separation:*
- *Drag of bluff body is due mainly to pressure distribution (pressure drag). However, this is not to mean that wall shear stress is not important. Recall that wall shear stress causes the large-scale separation in the first place.*
- *Certainly, there are the In-Betweens.*

1.8 AXIOM

The foregoing may now be generalized into the following simple axiom: "Acceleration of real fluids tends to be an efficient process, deceleration an inefficient one".

Accelerated motion occurs along the surface of front part of submerged object. (1.U is accompanied by a favorable pressure gradient which serves to stabilize the boundary layer and thus minimize energy dissipation. Decelerated motion is

accompanied by an adverse pressure gradient which tends to promote separation, instability, eddy formation and large energy dissipation.

1.9 Secondary Flow

Another consequence of wall friction is the creation of a flow within a flow - a ***secondary flow*** *superposed on the main primary flow.*

Fig.21 shows a cross section through a river bend. At the mean line fluid particles are in equilibrium under the centrifugal and pressure force. Near walls the pressure forces dominate the centrifugal forces. And considering the application of Bernoulli's equation, one expect that the flow velocity at the outer side is lower than that at the inner side with a pressure distribution opposite to the velocity distribution.

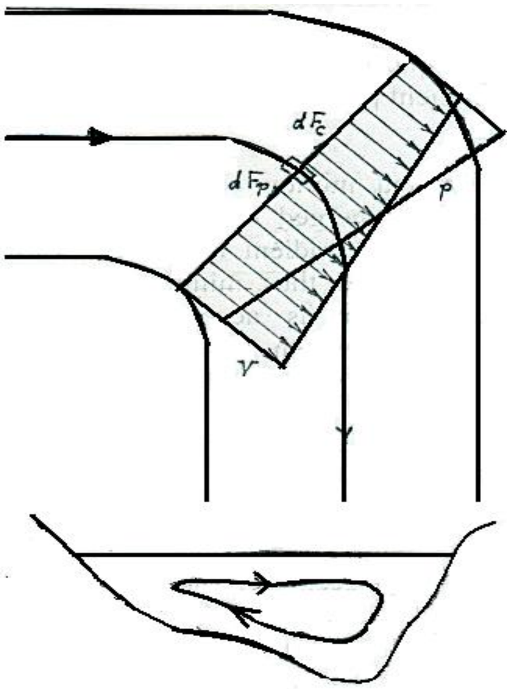


Fig.22

Therefore is originated the flow shown, which called secondary flow. The secondary flow serves to deposit material at the inside of the bend and to assist in scouring the outer side, thus producing the well-known meandering characteristics of natural stream.

Another example of secondary flow is shown in Fig. 22, where shown the horseshoe-shaped vortex that is produced by a projection from a boundary surface. The velocity profile along the wall leads to stagnation pressure at a being larger than that at B. This pressure difference maintains a downward secondary How from A to B, thus inducing a vortex-type of motion, the core of the vortex being swept downstream around the sides of the projection. This principle is used on some aircraft wings, the devices (called vortex generators) being used to draw higher energy fluid down to the wing surface to forestall large-scale separation.

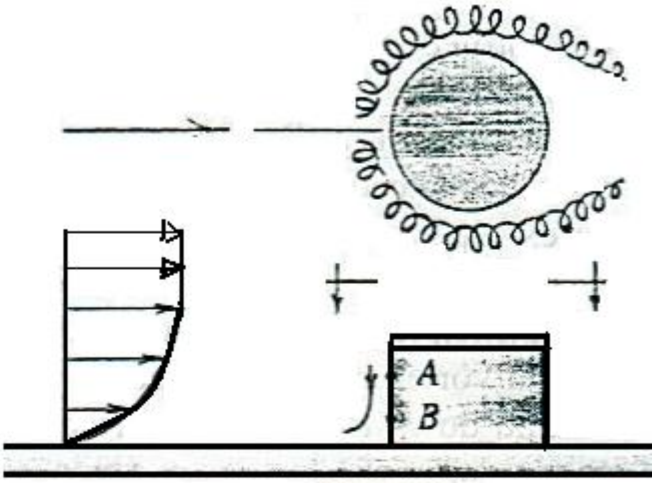


Fig. 22

SHEET (1)

"BOUNDARY LAYER THEORY"

1. *The velocity distribution in the boundary layer is given by* $\frac{u}{U} = 2\left(\frac{y}{\delta}\right) - \left(\frac{y}{\delta}\right)^2$ *,* δ *being boundary layer thickness. Calculate the following:*

    a) *Displacement thickness,*

    b) *Momentum thickness, and*

    c) *Energy thickness.*

2. *If velocity distribution in laminar boundary layer over a flat plate is assumed to be given by second order polynomial* $u = a + by + cy^2$*, determine its form using the necessary boundary conditions.*

3. *The velocity distribution in the boundary layer is given by* $\frac{u}{U} = \left(\frac{u}{\delta}\right)^{\frac{1}{7}}$ *Calculate the following:*

    a) *Displacement thickness,*

    b) *Momentum thickness,*

    c) *Shape factor*

    d) *Energy thickness and,*

    e) *Energy loss due to boundary layer if at a particular section, the boundary layer thickness is 25 mm and the free stream velocity is 15 m/s. If the discharge through the boundary layer region is 6* m^3/s *per meter width, express this energy loss in terms of meters of head. Take* $\rho = 1.2\ kg/m^3$*.*

4. *In the boundary layer over the face of a high spillway, the velocity distribution was observed to have the following form:*

$$\frac{u}{U} = \left(\frac{y}{\delta}\right)^{0.22}$$

The free stream velocity U is 20 m/s and boundary layer thickness 5 cm at a certain section. The discharge is 5 m^3/s per meter length of spillway. Calculate displacement thickness, energy thickness and loss of energy up to section under consideration.

5. *The free stream velocity U at a certain section was observed to be 30 m/s and a boundary layer thickness of 60 mm was estimated from the velocity distribution measured at the section. The discharge passing over the spillway was 6 m^3/s per meter length of spillway. Calculate:*

   a) *The displacement thickness,*

   b) *The energy thickness, and*

   c) *The loss of energy up to the section under consideration.*

6. *A smooth plate 2 m wide and 2.5 m long is towed in oil (sp. gr. = 0.8) at a velocity of 1.5 m/s along its length. Find the thickness of boundary layer and shear stress at the trailing edge of the plate.* $\nu_{oil} = 10^{-4}\, m^2/s.$

7. *A plate 450 mm x 150 mm has been placed longitudinally in a stream of crude oil (specific gravity 0.925 and kinematic viscosity of 0.9 stoke) which flows with velocity of 6 m/s. Calculate:*

   a) *The friction drag on the plate,*

   b) *Thickness of the boundary layer at the trailing edge, and*

   c) *Shear stress at the trailing edge.*

8. *Air is flowing over a flat plate 5 m long and 2.5 m wide with a velocity of 4 m/s at 15°C. If* $\rho = 1.208\ kg/m^3$ *and* $\nu = 1.47 \times 10^{-5}\ m^2/s$*, calculate:*

   a) *Length of plate over which the boundary layer is laminar, and thickness of the boundary layer (laminar),*

   b) *Shear stress at the location where boundary layer ceases to be laminar, and*

c) *Total drag force on both sides on that portion of plate where boundary layer is laminar.*

9. *Atmospheric air at 20°C is flowing parallel to a flat plate at a velocity of 2.8 m/s. Assuming cubic velocity profile and using exact Blasius solution estimate the boundary layer thickness and the local co efficient of drag (or skin friction) at x = 1.2 m from the leading edge of the plate, Also find the deviation of the approximate solution from the exact solution. Take the kinematic viscosity of air at 20°C=15.4 x 10^{-6} m^2/s.*

10. *Airflows over a plate 0.5 m long and 0.6 wide with a velocity of 4 m/s. The velocity profile is in the form* $\frac{u}{U} = \sin\left(\frac{\pi}{2}\frac{y}{\delta}\right)$ *If ρ = 1.24 kg/m^3 and ν = 0.15 x 10 $^{-4}$ m^2/s, calculate: (i) Boundary layer thickness at the end of the plate, (ii) Shear stress at 250 mm from the leading edge, and (iii) Drag force on one side of the plate.*

11. *Find the ratio of friction drag on the front half and rear half of the flat plate kept at zero incidence in a stream of uniform velocity, if the boundary layer is laminar over the whole plate.*

12. *Air at standard conditions is flowing over a flat plate which is 1 m long and 0.3 m wide. The flow is uniform at the leading edge of the plate. The velocity profile in the boundary layer is assumed to* $\frac{u}{U} = 2\left(\frac{y}{\delta}\right) - \left(\frac{y}{\delta}\right)^2$ *as the free stream velocity is U = 30 m/s. Assume that the flow conditions are independent of Z Using control volume abcd, shown by dashed line, calculate the mass flow rate across the surface ab. [Density of air may be taken as 1.23 kg/m^3, refer to Fig. 1.1]*

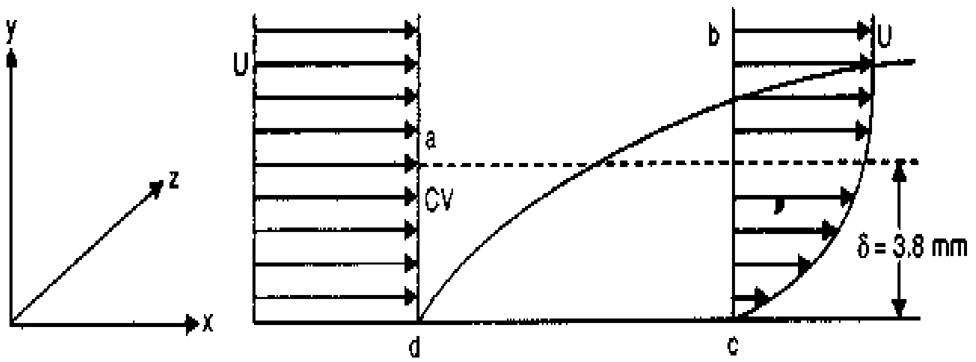


Fig. 1.1

DIMENSIONAL AND MODEL ANALYSIS

Introduction

Dimensional analysis is a mathematical technique which makes use of the study of the dimensions for solving several engineering problems.

Each physical phenomenon can be expressed by an equation giving relationship between different quantities; such quantities are dimensional and non-dimensional. Dimensional analysis helps in determining a systematic arrangement of the variables in the physical relationship, combining dimensional variables to form non-dimensional parameters. It is based on the principle of dimensional homogeneity and uses the dimensions of relevant variables affecting the phenomenon.

The method of dimensional analysis is used in every field of engineering, especially in such fields as fluid dynamics and thermodynamics where problems with many variables are handled. This method derives from the condition that each term summed in an equation depicting a physical relationship must have same dimension. By constructing non-dimensional quantities expressing the relationship among the variables, it is possible to summarize the experimental results and to determine their functional relationship.

Uses of dimensional analysis:

The uses of dimensional analysis may be summarized as follows:

1. *To test the dimensional homogeneity of any equation of fluid motion.*
2. *To derive rational formulae for a flow phenomenon.*
3. *To derive equations expressed in terms of non-dimensional parameters to show the relative significance of each parameter.*

4. *To plan model tests and present experimental results in a systematic manner; thus making it possible to analyze the complex fluid flow phenomenon.*

Advantages of dimensional analysis:

Dimensional analysis entails the following advantages:

1. *It expresses the functional relationship between the variables in .dimensionless terms.*
2. *in hydraulic model studies it reduces the number of variables involved in a physical phenomenon, generally by three.*
3. *By the proper selection of variables, the dimensionless parameters can be used to make certain logical deductions about the problem.*
4. *Design curves, by the use of dimensional analysis, can be developed from experimental data or direct solution of the problem.*
5. *It enables getting up a theoretical equation in a simplified dimensional form.*
6. *Dimensional analysis provides partial solutions to the problems that are too complex to be dealt with mathematically.*
7. *The conversion of units of quantities from one system to another is facilitated.*

2. Dimensions

The various physical quantities used in fluid phenomenon can be expressed in terms of ***fundamental quantities*** *or primary quantities. The fundamental quantities are mass, length, time and temperature, designated by the letters M, L, T, θ respectively. Temperature is especially useful in compressible flow. The quantities which are expressed in terms of the fundamental or primary quantities are called* ***derived or secondary quantities,*** *(e.g., velocity, area, acceleration etc.). The expression for a derived quantity in terms of the primary quantities is called the*

***dimension** of the physical quantity.*

*A quantity may either be expressed dimensionally in **M-L-T or F-L-T** system (some engineers prefer to use force instead of mass as fundamental quantity because the force is easy to measure). The table-1 gives the dimensions of various quantities used in both the systems.*

Fluid Mechanics and Heat Transfer and their Dimensions

Quantities used in Fluid Mechanics and Heat Transfer and their Dimensions

S. No.	Quantity	*Dimensions* *M-L-T System*	*F.L.T. System*
	(a) Fundamental Quantities		
1.	Mass, *M*	M	$FL^{-1}T^2$
2.	Length, *L*	L	L
3.	Time, *T*	T	T
	(b) Geometric Quantities		
4.	Area, *A*	L^2	L^2
5.	Volume, $\forall$	L^3	L^3
6.	Moment of inertia	L^4	L^4
	(c) Kinematic Quantities		
7.	Linear velocity, *u, V, U*	LT^{-1}	LT^{-1}
8.	Angular velocity, ω; rotational speed, *N*	T^{-1}	T^{-1}
9.	Acceleration, *a*	LT^{-2}	LT^{-2}
10.	Angular acceleration, α	T^{-2}	T^{-2}
11.	Discharge, *Q*	L^3T^{-1}	L^3T^{-1}
12.	Gravity, *g*	LT^{-2}	LT^{-2}
13.	Kinematic viscosity, ν	L^2T^{-1}	L^2T^{-1}
14.	Stream function, Ψ; circulation, Γ	L^2T^{-1}	L^2T^{-1}
15.	Vorticity, Ω	T^{-1}	T^{-1}
	(d) Dynamic Quantities		
16.	Force, *F*	MLT^{-2}	F
17.	Density, ρ	ML^{-3}	$FL^{-4}T^2$
18.	Specific weight, *w*	$ML^{-2}T^{-2}$	FL^{-3}
19.	Dynamic viscosity, μ	$ML^{-1}T^{-1}$	$FL^{-2}T$
20.	Pressure, *p*; shear stress, τ	$ML^{-1}T^{-2}$,	FL^{-2}
21.	Modulus of elasticity, *E, K*	$ML^{-1}T^{-2}$	FL^{-2}
22.	Momentum	MLT^{-1}	FT
23.	Angular momentum or moment of momentum	ML^2T^{-1}	FLT
24.	Work, *W*; energy, *E*	ML^2T^{-2}	FL
25.	Torque, *T*	ML^2T^{-2}	FL
26.	Power, *P*	ML^2T^{-3}	FLT^{-1}
	(e) Thermodynamic Quantities		
27.	Temperature	θ	θ
28.	Thermal conductivity	$MLT^{-3}\theta^{-1}$	$FT^{-1}\theta^{-1}$
29.	Enthalpy per unit mass	L^2T^{-2}	L^2T^{-2}
30.	Gas constant	$L^2T^{-2}\theta^{-1}$	$L^2T^{-2}\theta^{-1}$
31.	Entropy	$ML^2T^{-2}\theta^{-1}$	$FL\theta^{-1}$
32.	Internal energy per unit mass	L^2T^{-2}	L^2T^{-2}
33.	Heat transfer	ML^2T^{-3}	FLT^{-1}

Example.3-1. Determine the dimensions of the following quantities:
(i) Discharge, (ii) Kinematic viscosity,
(iii) Force and (iv) Specific weight.

Sol. (*i*) *Discharge* = area × velocity

$$= L^2 \times \frac{L}{T} = \frac{L^3}{T} = L^3 T^{-1} \text{ (Ans.)}$$

(*ii*) *Kinematic viscosity* $(\nu) = \frac{\mu}{\rho}$

where μ is given by : $\tau = \mu \frac{du}{dy}$

$$\mu = \frac{\tau}{\frac{du}{dy}} = \frac{\text{shear stress}}{\frac{L}{T} \times \frac{1}{L}} = \frac{\text{force/area}}{1/T}$$

$$= \frac{\text{mass} \times \text{acceleration}}{\text{area} \times 1/T} = \frac{M \times \frac{L}{T^2}}{L^2 \times \frac{1}{T}} = \frac{ML}{L^2T^2 \times \frac{1}{T}}$$

$$= \frac{M}{LT} = ML^{-1}T^{-1} \quad \text{and} \quad \rho = \frac{\text{mass}}{\text{volume}} = \frac{M}{L^3} = ML^{-3}$$

$$\therefore \text{ Kinematic viscosity } (\nu) = \frac{\mu}{\rho} = \frac{ML^{-1}T^{-1}}{ML^{-3}} = L^2T^{-1} \text{ (Ans.)}$$

(*iii*) *Force* = mass × acceleration

$$= M \times \frac{\text{length}}{\text{time}^2} = \frac{ML}{T^2} = MLT^{-2} \text{ (Ans.)}$$

(*iv*) *Specific weight* $= \frac{\text{weight}}{\text{volume}} = \frac{\text{force}}{\text{volume}} = \frac{MLT^{-2}}{L^3} = ML^{-2}T^{-2}$ **(Ans.)**

3-3. Dimensional Homogeneity

A physical equation is the relationship between two or more physical quantities. Any correct equation expressing a physical relationship between quantities must be dimensionally homogeneous (according to Fourier's principle of dimensional homogeneity) and, numerically equivalent. Dimensional homogeneity states that every term in an equation when reduced to fundamental dimensions must contain identical powers of each dimension; A dimensionally homogeneous equation is applicable to all systems of units. In a dimensionally homogeneous equation, only quantities having the same dimensions can be added, subtracted or equated. Let us consider the equation: $\boldsymbol{P = \gamma \cdot h}$

Dimensions of L.H.S. $= ML^{-1}T^{-2}$

Dimensions of R.H.S. $= ML^{-2}T^{-2} \times L = ML^{-1}T^{-2}$

Dimensions of L.H.S. = dimensions of R.H.S.

*Equation **P= γ.h** is dimensionally homogeneous; so it can be used in any system of units.*

Applications of Dimensional Homogeneity:

The principle of homogeneity proves useful in the following ways:

1. *It facilitates to determine the dimensions of a physical quantity.*
2. *It helps to check whether an equation of any physical phenomenon is dimensionally homogeneous or not.*
3. *It facilitates conversion of units from one system to another.*
4. *It provides a step towards dimensional analysis which is fruitfully employed to plan experiments and to present the results meaningfully.*

3-4. Methods of Dimensional Analysis

With the help of dimensional analysis the- equation of a physical phenomenon can be developed in terms of dimensionless groups or parameters and thus reducing the number of variables. The methods of dimensional analysis are based on the Fourier's principle of homogeneity. The methods of dimensional analysis are:

1. ***Rayleigh's method***
2. ***Buckingham's π- method***
3. ***Bridgman's method***
4. ***Matrix-tensor method***
5. ***By visual inspection of the variables involved***
6. ***Rearrangement of differential equations.***

Here only first two methods will be dealt with.

3-4-1. Rayleigh's Method

This method gives a special form of relationship among the dimensionless group, and has the inherent drawback that it does not provide any information regarding the number of dimensionless groups to be obtained as a result of dimensional analysis. Due to this reason this method has become obsolete and is not favoured for use.

Rayleigh's method *is used for determining the expression for a variable which depends upon maximum three or four variables only. In case the number of independent variables becomes more than four, then it is very difficult to find the expression for the dependent variable.*

In this method a functional relationship of some variables is expressed in the form of an exponential equation which must be dimensionally homogeneous. Thus if X is a variable which depends on X_t, X_2, X_3 ...Xn, the functional equation can be written as:

$$X=f\,(X_1, X_2, X_3,\ldots\ldots X_n) \quad \ldots\ldots\ldots\ldots\ldots 3\text{-}1$$

In the above equation X is a dependent variable, while $X_,$, X_2, X_3, ... X_n are independent variables. A dependent variable is the one about which information is required while independent variables are those which govern the variation of dependent variable. Eqn. (3-1) can also be written as:

$$x=(x_1^a, x_2^b, x_3^c, \ldots\ldots x_n^n) \quad \ldots\ldots\ldots\ldots 3\text{-}2$$

Where C is a constant and a, b, c,...., n are the arbitrary powers. The values of a, b, c, ... n are obtained by comparing the powers of the fundamental dimensions on both sides. Thus the expression is obtained for dependent variable

Example 3.2*: Find an expression for the drag force on smooth sphere of diameter D, moving with a uniform velocity V in a fluid density ρ and dynamic viscosity μ.*

Sol. The drag force F is a function of

(*i*) Diameter D, (*ii*) Velocity V,

(*iii*) Fluid density ρ, and (*iv*) Dynamic viscosity μ.

Mathematically, $F = f(D, V, \rho, \mu)$ or $F = C\,(D^a \,.\, V^b \,.\, \rho^c \,.\, \mu^d)$ ...(1)

where C is a non-dimensional constant.

Using M–L–T system the corresponding equation for dimensions is:

$$MLT^{-2} = [C\,L^a \,.\, (LT^{-1})^b \,.\, (ML^{-3})^c \,.\, (ML^{-1}T^{-1})^d]$$

For dimensional homogeneity the exponents of each dimension on both sides of the equation must be identical. Thus

For M : $1 = c + d$ ...(*i*)

For L : $1 = a + b - 3c - d$ ...(*ii*)

For T : $-2 = b - d$ ...(*iii*)

There are *four unknowns* (a, b, c, d) but *equations are three* in number. Therefore, it is not possible to find the values of a, b, c and d. However, three of them can be expressed in terms of fourth variable which is most important. There the role of viscosity is vital one and hence a, b, c are expressed in terms of d (*i.e.* power to viscosity)

$\therefore$ $c = 1 - d$ ...from (*i*)

$b = 2 - d$ ...from (*iii*)

Putting these values in (*i*), we get

$$a = 1 - b + 3c + d = 1 - 2 + d + 3\,(1 - d) + d$$
$$= 1 - 2 + d + 3 - 3d + d = 2 - d$$

Substituting these values of exponents in eqn. (1), we get

$$F = C\,[D^{2-d} \,.\, V^{2-d} \,.\, \rho^{1-d} \,.\, \mu^d]$$
$$= C\left[D^2 V^2 \rho\,(D^{-d} \,.\, V^{-d} \,.\, \rho^{-d} \,.\, \mu^d)\right] = C\left[\rho D^2 V^2 \left(\frac{\mu}{\rho V D}\right)^d\right]$$
$$= \rho D^2 V^2 \phi\left(\frac{\mu}{\rho V D}\right) \textbf{(Ans.)}$$

2. Buckingham's π-Method/Theorem

When a large number of physical variables are involved Rayleigh's method of dimensional analysis becomes increasingly laborious and cumbersome. Buckingham's method is an improvement over the Rayleigh's method. Buckingham designated the dimensionless group by the Greek capital letter ***π*** *(Pi). It is therefore often called Buckingham* ***π*** *-method. The advantage of this method over Rayleigh's method is that it let us know, in advance of the analysis, as to how many dimensionless groups are to be expected.*

The Buckingham's re-theorem states as follows:

*"If there are **n** variables (dependent and independent variables) in a dimensionally homogeneous equation and if these variables contain **m** fundamental dimensions (such, as **M, L, T**, etc.), then the variables are arranged into **(n-m)** dimensionless terms. These dimensionless terms are called **π** -terms.*

Mathematically, if any variable X_1, depends on independent variables, $X_2, X_3, X_4, \ldots X_n$ the functional equation may be written as

$$X_L = f(X_2,\ X3,\ X4 \ldots X_n) \qquad \ldots (3\text{-}3)$$

Eqn. (3-3) can also be written as

$$f = (X_1, X_2, X_3 \ldots Xn) = 0 \qquad \ldots (3\text{-}4)$$

*It is a dimensionally homogeneous equation and contains n variables. If there are m fundamental dimensions then according to Buckingham's **π**-theorem, it [eqn. (3-4)] can be written in terms of number of **π**-terms (dimensionless groups) in which number of **π** -terms is equal to (n-m). Hence eqn. (3-4) becomes as*

$$f_1(\pi 1, \pi_2, \pi_3, \ldots \pi_{n-m}) = 0 \qquad \ldots (3\text{-}5)$$

*Each dimensionless **π** -term is formed by combining m variables out of the total n variables with one of the remaining (n-m) variables i.e. each **π** -terms contain (m+ 1) variables. These m variables which appear repeatedly in each of **π**-terms are consequently called repeating variables and are chosen from among the variables such that they together involve all the fundamental dimensions and they themselves do not form a dimensionless parameter. Let in the above case X_2, X_3, and X_4 are the repeating variables if the fundamental dimensions m (M, L, T) =3. Then each term is written as:*

$$\left.\begin{aligned} \pi_1 &= X_2^{a_1} \cdot X_3^{b_1} \cdot X_4^{c_1} \cdot X_1 \\ \pi_2 &= (X_2^{a_2} \cdot X_3^{b_2} \cdot X_4^{c_2} \cdot X_5) \\ &\vdots \\ \pi_{n-m} &= (X_2^{a_{n-m}} \cdot X_3^{b_{n-m}} \cdot X_4^{c_{n-m}} \cdot X_n) \end{aligned}\right\} \quad \ldots\ldots(3.6)$$

where a_1- b_1- c_1 ; a_2 - b_2 - c_2 etc. are the constants, which are determined, by considering dimensional homogeneity. These values are substituted in eqn. (3-6) and values of $\pi_1, \pi_2, \pi_3, \ldots \pi_{n-m}$ are obtained. These values of π's are substituted in eqn. (3-5). The, final general equation for the phenomenon may then be obtained by expressing anyone of the π-terms as a function of the other as

$$\left.\begin{aligned} \pi_1 &= \phi(\pi_2, \pi_3, \pi_4, \ldots \pi_{n-m}) \\ \pi_2 &= \phi(\pi_1, \pi_3, \pi_4, \ldots \pi_{n-m}) \end{aligned}\right\} \qquad \ldots\ldots(3.7)$$

Selection of repeating variables:

The following points should be kept in view while selecting m repeating variables:

1. m repeating variables must contain jointly all the fundamental dimensions involved in the phenomenon. Usually the fundamental dimensions are M, L and T. However, if only two dimensions are involved, there will be 2 repeating variables and they must contain together the two dimensions involved.

2. The repeating variables must not form the non-dimensional parameters among themselves.

3. As far as possible, the dependent variable should not be selected as repeating variable.

4. No two repeating variables should have the same dimensions.

5. The repeating variables should be chosen in such a way that one variable contains geometric property (e.g. length, I; diameter, d; height, H etc.), other variable contains flow property (e.g. velocity, V; acceleration, a etc.) and third variable contains fluid property (e.g. mass density ρ; weight density γ, dynamic viscosity μ, etc.). The choice of repeating variables, in most of fluid mechanics problems, may be:

(i) l, V, ρ (ii) d, V, ρ

(iii) l, V, μ (iv) d, V, μ.

The procedure for solving problems by Buckingham's π-theorem is outlined in the Example 3-4 below:

Example 3-4. The resistance R experienced by a partially submerged body depends upon the velocity V, length of the body l, viscosity of the fluid μ, density of the fluid ρ and gravitational acceleration g . Obtain a dimensionless expression for R.

Sol, Step. 1. The resistance R is a function of:

(i) Velocity V, *(ii) Length l*

(iii) Viscosity μ, and *(iv) Density ρ,*

(v) Gravitational acceleration g.

Mathematically, $R = f(V, l, \mu, \rho, g)$ *or* $f_1(R, V, l, \mu, \rho, g) = 0$

Total number of variables, n = 6, m is obtained by writing dimensions of each variable as:

$$\mathbf{R = MLT^{-2},\ V = LT^{-1},\ \mu = ML^{-1}T^{-1},\ \rho = ML^{-3},\ g = LT^{-2}}.$$

*Thus the fundamental dimensions in the problem are **M, L, T** and hence **m=3**.*

*Number of dimensionless **Pi-π- terms=n - m=6-3=3.***

*Thus three **π-terms** say π_1, π_2, π_3 are formed.*

The eqn. may be written as $f_1(\pi_1, \pi_2, \pi_3) = 0$

Step 2: Selection of repeating variables: *Out of six variables R, V, l, μ, ρ, g three variables **(as m = 3)** are to be selected as repeating variables. **R** is a dependent variable and should not be selected as a repeating variable. Out of the remaining five variables one variable should have geometric property, second should have flow property and third one should have fluid property; these requirements are met by selecting **l, V** and **ρ** as repeating variables. The repeating variables themselves should not form a dimensionless term and must contain jointly all fundamental dimensions equal to **m** i.e. **3** here. Dimensions of **l, V** and ρ are* $\mathbf{L,\ LT^{-1}, ML^{-3}}$ *and hence the three fundamental dimensions exist in **l, V** and **ρ** and also no dimensionless group is formed by them.*

Step 3: Each π -term (= m + 1 variable) is written as given in eqn. (3-6), i.e.

$$\left.\begin{aligned}\pi_1 &= l^{a_1}\,.\,V^{b_1}\,.\,\rho^{c_1}\,.\,R\\ \pi_2 &= l^{a_2}\,.\,V^{b_2}\,.\,\rho^{c_2}\,.\,\mu\\ \pi_3 &= l^{a_3}\,.\,V^{b_3}\,.\,\rho^{c_3}\,.\,g\end{aligned}\right\}$$

Step 4: Each π -term is solved by the principle of dimensional homogeneity, as follows:

π_1–term :

$$\pi_1 = l^{a_1}\,.\,V^{b_1}\,.\,\rho^{c_1}\,.\,R$$

$$M^0L^0T^0 = L^{a_1}\,.\,(LT^{-1})^{b_1}\,.\,(ML^{-3})^{c_1}\,.\,(MLT^{-2})$$

Equating the exponents of M, L and T respectively, we get

For M : $0 = c_1 + 1$

For L : $0 = a_1 + b_1 - 3c_1 + 1$

For T : $0 = -b_1 - 2$

$\therefore$ $c_1 = -1$; $b_1 = -2$

and $a_1 = -b_1 + 3c_1 - 1 = 2 - 3 - 1 = -2$

Substituting the values of a_1, b_1, and c_1 in π_1, we get

$$\therefore \quad \pi_1 = l^{-2}\,.\,V^{-2}\,.\,\rho^{-1}\,.\,R = \frac{R}{l^2V^2\rho}$$

π_2–term:

$$\pi_2 = l^{a_2}\,.\,V^{b_2}\,.\,\rho^{c_2}\,.\,\mu$$

$$M^0L^0T^0 = L^{a_2}\,.\,(LT^{-1})^{b_2}\,.\,(ML^{-3})^{c_2}\,.\,(ML^{-1}T^{-1})$$

Equating the exponents of M, L and T respectively, we get

For M : $0 = c_2 + 1$

For L : $0 = a_2 + b_2 - 3c_2 - 1$

For T : $0 = -b_2 - 1$

$\therefore$ $c_2 = -1$; $b_2 = -1$

and $a_2 = -b_2 + 3c_2 + 1 = 1 - 3 + 1 = -1$

$$\therefore \qquad \pi_2 = l^{-1} \cdot V^{-1} \cdot \rho^{-1} \cdot \mu = \frac{\mu}{lV\rho}$$

π_3—term:

$$\pi_3 = l^{a_3} \cdot V^{b_3} \cdot \rho^{c_3} \cdot g$$

$$M^0L^0T^0 = L^{a_3} \cdot (LT^{-1})^{b_3} \cdot (ML^{-3})^{c_3} \cdot (LT^{-2})$$

Equating the exponents of M, L and T respectively, we get

For M : $0 = c_3$

For L : $0 = a_3 + b_3 - 3c_3 + 1$

For T : $0 = -b_3 - 2$

$\therefore \qquad c_3 = 0 \; ; \; b_3 = -2$

and $\qquad a_3 = -b_3 + 3c_3 - 1 = 2 + 0 - 1 = 1$

Substituting the values of a_3, b_3, and c_3 in π_3, we get

$$\therefore \qquad \pi_3 = l^1 \cdot V^{-2} \cdot \rho^0 \cdot g = \frac{lg}{V^2}$$

Step 5: Substitute the values of π_1, π_2, π_3. The functional relationship becomes

$$f_1\left(\frac{R}{l^2V^2\rho}, \frac{\mu}{lV\rho}, \frac{lg}{V^2}\right) = 0$$

$$\frac{R}{l^2V^2\rho} = \phi\left(\frac{\mu}{lV\rho}, \frac{lg}{V^2}\right)$$

$$= \phi\left(\frac{\rho V l}{\mu}, \frac{V}{\sqrt{lg}}\right)$$

The above step has been made on the postulate that reciprocal of pi-term and its square root is non-dimensional.

$$R = l^2V^2\rho\,\phi\left(\frac{\rho V l}{\mu}, \frac{V}{\sqrt{lg}}\right)$$

The resistance R is thus a function of Reynolds number

$$\left(\frac{\rho V l}{\mu}\right),$$

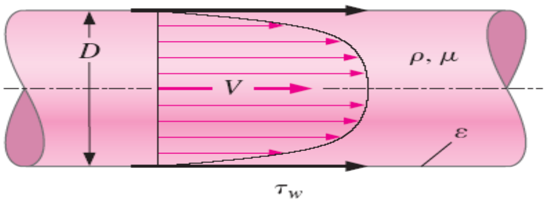
and Froude's number

$$\left(\frac{V}{\sqrt{lg}}\right)$$

EXAMPLE 3-5: Friction in a Pipe

- *Consider flow shown in Fig.; V is the average speed across the pipe cross section. The flow is fully developed, which means that the velocity profile also remains uniform down*

the pipe. Because of frictional forces between the fluid and the pipe wall, there exists a shear stress tw on the inside pipe wall. The shear stress is also constant down the pipe in the region. We assume some constant average roughness height, ε, along the inside wall of the pipe. In fact, the only parameter that is not constant down the length of pipe is the pressure, which must decrease (linearly) down the pipe in order to "push" the fluid through the pipe to overcome friction. Develop a nondimensional relationship between shear stress t_w and the other parameters in the problem.



Solution

List of relevant parameters: $\tau_w = f(V, \varepsilon, \rho, \mu, D) \qquad n = 6$

primary dimensions

τ_w	V	ε	ρ	μ	D
$\{m^1L^{-1}t^{-2}\}$	$\{L^1t^{-1}\}$	$\{L^1\}$	$\{m^1L^{-3}\}$	$\{m^1L^{-1}t^{-1}\}$	$\{L^1\}$

Reduction: $j = 3 \quad \Rightarrow \quad k = n - j = 3$

Repeating parameters: V, D, and ρ

$$\Pi_1 = \tau_w V^{a_1} D^{b_1} \rho^{c_1} \quad \rightarrow \quad \Pi_1 = \frac{\tau_w}{\rho V^2}$$

Modified Π_1: $$\Pi_{1,\text{ modified}} = \frac{8\tau_w}{\rho V^2} = \text{Darcy friction factor} = f$$

$$\Pi_2 = \mu V^{a_2} D^{b_2} \rho^{c_2} \quad \rightarrow \quad \Pi_2 = \frac{\rho V D}{\mu} = \text{Reynolds number} = \text{Re}$$

$$\Pi_3 = \varepsilon V^{a_3} D^{b_3} \rho^{c_3} \quad \rightarrow \quad \Pi_3 = \frac{\varepsilon}{D} = \text{Roughness ratio}$$

$$\Rightarrow \quad f = \frac{8\tau_w}{\rho V^2} = f\left(\text{Re}, \frac{\varepsilon}{D}\right)$$

3-4-3. Limitations of Dimensional Analysis

Following are the limitations of dimensional analysis:

1. *Dimensional analysis does not give any clue regarding the selection of variables. If the variables are wrongly taken, the resulting functional relationship is erroneous. It provides the information about the grouping of variables. In order to decide whether selected variables are pertinent or superfluous experiments have to be performed.*
2. *The complete information is not provided by dimensional analysis; it only indicates that there is some relationship between parameters. It does not give the values of co-efficient in the functional relationship. The values of co-efficient and hence the nature of functions can be obtained only from experiments or from mathematical analysis.*

MODEL ANALYSIS

Introduction

In order to know about the performance of the hydraulic structures (e.g. dams, spillways etc.) or hydraulic machines (e.g. turbines, pumps etc.) before actually constructing or manufacturing them, their models are made and tested to get the required information. The model is the small scale replica of the actual structure or machine. The actual structure or machine is called Prototype. The models are not always smaller than the prototype, in some cases a model may be even larger or of the same size as prototype depending upon the need and purpose

Advantages of model testing

The following are the advantages of model analysis:

1. The model tests are quite economical and convenient (because the design, construction and operation of a model may be changed several times if necessary, without increasing much expenditure, till most suitable design is obtained).

2. With the use of models the performance of hydraulic structures/hydraulic machines can be predicted in advance.

3. While designing a particular portion of the structure if clear cut analytical and reliable method is not available then in such cases it becomes absolutely necessary to know about the safety and reliability of such parts which is possible by means of model testing.

4. Model testing can be used to detect and rectify the defects of an existing structure which is not functioning properly.

Applications of the model testing

Following are the important fields where applications of the model testing is of great use:

1. Civil engineering structures such as dams, spillways, weirs, canals etc.

2. Scour in rivers, irrigation channels.

3. Turbines, pumps and compressors.

4. Design of ships and submarines.

5. Aeroplane, rockets and missiles.

6. Tall buildings (to predict the wind loads on buildings, the stability characteristics of the buildings and airflow patterns in their vicinity).

Next, in order to determine the characteristics of a full-scale device through model tests, besides geometrical similarity, similarity of dynamical conditions between the two is also necessary. When the above dimensional analysis is employed, if the appropriate non-dimensional quantities such as Reynolds number and Froude number are the same for both devices, the results of the model device tests are applicable to the full-scale device.

When the characteristics of a water wheel, pump, boat or aircraft are obtained by means of a model, unless the flow conditions are similar in addition to the shape, the characteristics of the prototype cannot be assumed from the model test result. In order to make the flow conditions similar, the respective ratios of the corresponding forces acting on the prototype and the model

should be equal.

7. Forces Influencing Hydraulic Phenomena (e.g.), *the forces acting on the flow element:*

The forces acting on the flow element are due to gravity F_G*, pressure* Fp*, viscosity* F_V*, surface tension* F_T *(when the prototype model is on the boundary of water and air), inertia* F_I*, and elasticity* F_E*. The forces can be expressed as shown below.*

The forces which may affect/influence the flow characteristics of a problem are:

1. Inertia force (F_I)

It always exists in the fluid flow problem (and hence it is customary to find out the force ratios with respect to inertia force). It is equal to the product of mass and acceleration of the flowing fluid and acts in the direction opposite to the direction of acceleration.

2. Viscous force (F_T)

It is present in fluid flow problems where viscosity is to play an important role. It is equal to the product of shear stress (τ*) due to viscosity and surface area of the flow.*

3. Gravity force (F_G)

It is present in case of open surface flow. It is equal to the product of mass and acceleration due to gravity,

4. Pressure force (F_P)

This type of force is present in case of pipe-flow. It is equal to the product of pressure intensity and cross-sectional area of the flowing fluid.

5. Surface tension force (F_S)

It is equal to the product of surface tension and length of surface of the flowing fluid.

6. Elastic force (F_E)

It is equal to the product of elastic stress and area of

the flowing fluid.

gravity force	$F_G = mg = \rho L^3 g$
pressure force	$F_P = (\Delta p)A = (\Delta p)L^2$
viscous force	$F_V = \mu\left(\frac{\mathrm{d}u}{\mathrm{d}y}\right)A = \mu\left(\frac{v}{L}\right)L^2 = \mu v L$
surface tension force	$F_T = TL$
inertial force	$F_I = m\alpha = \rho L^3 \frac{L}{T^2} = \rho L^4 T^{-2} = \rho v^2 L^2$
elasticity force	$F_E = KL^2$

Since it is not feasible to have the ratios of all such corresponding forces simultaneously equal, it will suffice to identify those forces that are closely related to the respective flows and to have them equal. In this way, the relationship which gives the conditions under which the flow is similar to the actual flow in the course of a model test is called the law of similarity. In the following section, the more common force ratios which ensure the flow similarity under appropriate conditions are developed.

6. Similarity

To find solutions to numerous complicated problems in hydraulic engineering and fluid mechanics model studies are usually conducted. In order that results obtained in the model studies represent the behavior of prototype, the following three similarities must be ensured between the model and the prototype.

1. Geometric similarity;

2. Kinematic similarity, and

3. Dynamic similarity.

1. Geometric similarity:

For geometric similarity to exist between the model and the prototype the ratios of corresponding lengths in the model and in the prototype must be same and the included angles between two corresponding sides must be the same. Models which are not geometrically similar are known as geometrically distorted models.

Let L_m = length of model,

H_m = height of model,

D_m = diameter of model,

A_m = area of model,

V_m = volume of model,

and L_p, B_p, H_p, D_p, A_p and V_p = corresponding values of the prototype.

Then, for *geometric similarity*, we must have the relation:

$$\frac{L_m}{L_p} = \frac{B_m}{B_p} = \frac{H_m}{H_p} = \frac{D_m}{D_p} = L_r$$

where L_r is called the *scale ratio* or the scale factor.

Similarly $$A_r = \text{area ratio} = \frac{A_m}{A_p} = L_r^2$$

and $$V_r = \text{volume ratio} = \frac{V_m}{V_p} = L_r^3$$

2. Kinematic similarity:

Kinematic similarity is the similarity of motion. If at the corresponding points in the model and in the prototype, the velocity or acceleration ratios are same and velocity or acceleration vectors point in the same direction, the two flows are said to be kinematically similar.

$$\frac{(V_1)_m}{(V_1)_p} = \frac{(V_2)_m}{(V_2)_p} = V_r, \text{ velocity ratio}$$

Similarly $$\frac{(a_1)_m}{(a_1)_p} = \frac{(a_2)_m}{(a_2)_p} = a_r, \text{ acceleration ratio}$$

The directions of the velocities in the model and prototype should be same.The geometric similarity is a pre-requisite for kinematic similarity.

3. Dynamic similarity:

Dynamic similarity is the similarity of forces. The flows in the model and in prototype are dynamically similar if at all the corresponding points, identical types of forces are parallel and bear the same ratio. In dynamic similarity, the force polygons of the two flows can be superimposed by change in force scale.,

Let $(F_i)_m$ = inertia force at a point in the model,

$(F_v)_m$ = viscous force at the point in the model,

$(F_g)_m$ = gravity force at the point in the model,

and $(F_i)_p, (F_v)_p, (F_g)_p$ = corresponding values of forces at the corresponding points in prototype.

Then for *dynamic similarity*, we have

$$\frac{(F_i)_m}{(F_i)_p} = \frac{(F_v)_m}{(F_v)_p} = \frac{(F_g)_m}{(F_g)_p} \ldots\ldots = F_r, \text{ force ratio}$$

The directions of the corresponding forces at the corresponding points in the model and prototype should also be same.

8. Dimensionless Numbers and their Significance

The dimensionless numbers (also called non-dimensional parameters) are obtained by dividing the inertia force (which always exists when any mass in motion) by viscous force or gravity force or pressure force or surface tension force or elastic force. The important dimensionless numbers:

1. Reynolds Number Re: *It is defined as the ratio of the inertia force to the viscous force.*

2. Froude number (Fr): *It is defined as the square root of the ratio of the inertia force and the gravity force.*

3. Euler's Number (Eu): *It is defined as the square root of the ratio of the inertia force to the pressure force.*

4. Weber Number (We): *It is defined as the square root of the ratio of the inertia force to the surface tension force.*

5. Mach Number (M): *It is defined as the square root of the ratio of the inertia force to the elastic force.*

Table : Dimensionless groups/numbers

Sl. No.	Dimensionless numbers	Aspects			
		Symbol	*Group of variables*	*Significance*	*Field of application*
1.	*Reynolds number*	*Re*	$\frac{\rho VL}{\mu}$	$\frac{\text{Inertia force}}{\text{Viscous force}}$	Laminar viscous flow in confined passages (where *viscous effects are significant*)
2.	*Froude's number*	*Fr*	$\frac{V}{\sqrt{Lg}}$	$\frac{\text{Inertia force}}{\text{Gravity force}}$	Free surface flows (where *gravity effects are important*)
3.	*Euler's number*	*Eu*	$\frac{V}{\sqrt{p/\rho}}$	$\frac{\text{Inertia force}}{\text{Pressure force}}$	Conduit flow (where *pressure variations are significant*)
4.	*Weber's number*	*We*	$\frac{V}{\sqrt{\sigma/\rho L}}$	$\frac{\text{Inertia force}}{\text{Surface tension}}$	Small surface waves, capillary and sheet flow (where *surface tension is important*)
5.	*Mach's number*	*M*	$\sqrt{\frac{V}{K/\rho}}$	$\frac{\text{Inertia force}}{\text{Elastic Force}}$	High speed flow (where *compressibility effects* are *significant*).

Similarity Laws

To ensure dynamic similarity between the model and prototype it is necessary that the ratio of the corresponding forces acting at the corresponding points in the model and prototype be made equal. It implies that dimensionless numbers should be same for the model as well as the prototype; this condition is difficult to be satisfied for all the dimensionless numbers. Hence models are designed on the basis of the force which is dominating in the flow situation. The laws on which the models are designed for dynamic similarity are called model or similarity laws; these are:

1. Reynolds model law,

2. Froude model law,

3. Weber model law, and

4. Mach model law.

5. Euler model law,

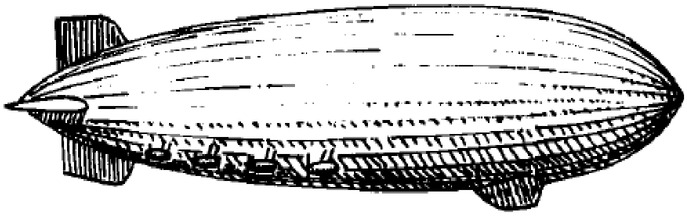
1. Reynolds model law,

In flow situations where in addition to inertia, viscous force is the only other predominant force. Where the compressibility of the fluid may be neglected and in the absence of a free surface, e.g. where fluid is flowing in a pipe, an airship is flying in the air (Fig.

3.2) or a submarine is navigating under water, only the viscous force and inertia force are of importance. The similarity of flow in the model and its prototype can be established if Reynolds number is same for both the systems.

This is known as Reynolds law and according to this law

Re Model = Re Prototype



$$\frac{\text{inertia force}}{\text{viscous force}} = \frac{F_I}{F_V} = \frac{\rho v^2 L^2}{\mu v L} = \frac{L v \rho}{\mu} = \frac{L v}{\nu} = Re$$
,

which defines the Reynolds number Re,

$$Re = Lv/\nu$$

Consequently, when the Reynolds numbers of the prototype and the model are equal the flow conditions are similar.

Following are some of the phenomena for which Reynolds model law can be a sufficient criterion for similarity of flow in the model and the prototype: (i) Motion of air planes,

(i) Flow of incompressible fluid in closed pipes,

(ii) Motion of submarines completely under water, and

(iii) Flow around structures and other bodies immersed completely under moving fluids

For similar dynamical flow conditions, the ratio of corresponding forces acting at corresponding points in the model and prototype should be equal. The ratios of forces are dimensionless numbers. It means for dynamic similarity between model and prototype, dimensionless numbers should be same for model and prototype.

Since the submarine has to overcome the viscous resistance, there has to be dynamic similarity between the model and the prototype; which implies equality of Reynolds number.

2. Froude model law,

When the resistance due to the waves produced by motion of a boat (gravity wave) is studied, the ratio of inertia force to gravity force ***is*** *important:*

$$\frac{\text{inertia force}}{\text{gravity force}} = \frac{F_I}{F_G} = \frac{\rho v^2 L^2}{\rho L^3 g} = \frac{v^2}{gL}$$

In general, in order to change ***v2*** *above to* ***v*** *as in the case for* ***Re,*** *the square root of* ***u2/gL*** *is used. This square root is defined as the Froude number* ***Fr,***

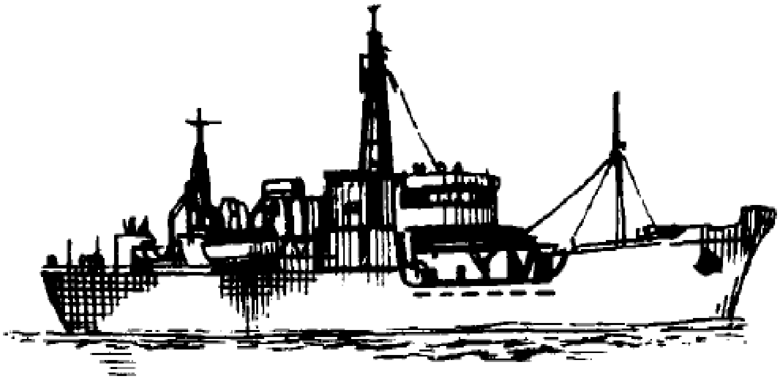
$$Fr = \frac{v}{\sqrt{gL}}$$

When the gravitational force can be considered to be the only predominant force which controls the motion in addition to the inertia force, the similarity of the flow in any two such systems can be established if the Froude's number for both the systems is the same. This is known as Froude Model Law.

Some of the phenomena for which the Froude model law can be sufficient criterion for dynamic similarity to be established in the model and the prototype are: (1) Free surface flows such as flow over spillways, sluices etc.; (2) Flow of jet from an orifice or nozzle; (3) Where waves are likely to be formed on the surface; (4) Where fluids of different mass densities flow over one another.

If a test is performed by making the ***Fr*** *of the actual boat (Fig.* ***3.3)*** *and of the model ship equal, the result is applicable to the actual boat so far as the wave resistance alone is concerned. This relationship is called Froude's law of similarity. For the total resistance, the frictional resistance must be taken into account in addition to the wave resistance. Also included in the circumstances where gravity inertia forces are important are flows in an open ditch, the force of water acting on a bridge pier, and flow running out of a water gate.*

$$Fr = \frac{v}{\sqrt{gL}}$$



3. Weber model law, and

When a moving liquid has its face in contact with another fluid or a solid, the inertia and surface tension forces are important:

$$\frac{\text{inertia force}}{\text{surface tension}} = \frac{F_I}{F_T} = \frac{\rho v^2 L^2}{TL} = \frac{\rho v^2 L}{T}$$

In this case, also, the square root is selected to be defined as the Weber number, ***We.***

$$We = v\sqrt{\rho L/T}$$

In a fluid system where surface tension effects predominate in addition to inertia force, the dynamic similarity is obtained by equating the Weber number for the model and its prototype, which is known as Weber model law.

According to this law:

$$We = v\sqrt{\rho L/T}$$

Weber model law is applied in the following flow situations:

Flow over weirs involving very low heads; Very thin sheet of liquid, flowing over a surface; Capillary waves in channels; Capillary rise in

narrow passages; Capillary movement of water in soil.

4. Mach model law.

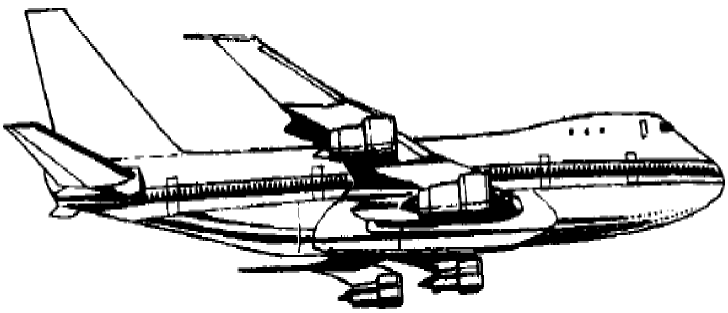
When a fluid flows at high velocity, or when a solid moves at high velocity in a *fluid at rest, the compressibility of the fluid can dominate so that the ratio of the inertia force to the elasticity force is then important.*

$$\frac{\text{inertia force}}{\text{elastic force}} = \frac{F_I}{F_E} = \frac{\rho v^2 L^2}{KL} = \frac{v^2}{K/\rho} = \frac{v^2}{a^2}$$

Again, in this case, the square root is selected to be defined as the Mach number ***M,***

$$M = v/a$$

M *< 1, M = 1 and M > 1 are respectively called subsonic flow, sonic flow and supersonic flow. When M = 1 and M < 1 and M > 1 zones are coexistent, the flow is called transonic flow.*



Boeing 747: full length, 70.5 m; full width, 59.6 m; passenger capacity, 498 persons; turbofan engine and cruising speed of 891 km/h (M = 0.82)

When in any fluid system only the forces resulting from elastic compression are significant in addition to inertial force, then the dynamic similarity between the model and its prototype may be achieved by equating the Mach numbers, which is known as Mach model law.

$$M = v/a$$

The similitude based on Mach model law finds application in the following:

(*i*) Aerodynamic testing;

(*ii*) Phenomena involving velocities exceeding the speed of sound;

(*iii*) Hydraulic model testing for the cases of unsteady flow, especially water hammer problems.

(*iv*) Under-water testing of torpedoes.

Example **(Model testing of ships)**. *A 1:20 model of a naval ship having a submerged area of 5 m^2 and length 8 metres has a total drag of 20 N when towed through water at a velocity of 1·5 m/s. Calculate the total drag on the prototype when moving at the corresponding speed. Use the relation $F_f = \frac{1}{2} C_f \rho A V^2$ for calculating the skin resistance. The value of C_f is given by, $C_f = 0{\cdot}0735/(Re)^{1/5}$.*

Take kinematic viscosity of water (or sea water) as 0·01 stoke and the specific weight of water (or sea water) as 9810 N/m^3

Sol. Linear scale ratio, $L_r = 20$

Submerged area of the model, $A_m = 5\ \mathrm{m}^2$

Length of the model, $L_m = 8$ m

Total drag of model, $R_m = 20$ N

Velocity of model, $V_m = 1{\cdot}5$ m/s

Let A_p, L_p, R_p, V_p be the corresponding values for the prototype.

Kinematic viscosity of sea water,

$$\nu_m = \nu_p = 0{\cdot}01 \text{ stokes} = 0{\cdot}01 \text{ cm}^2/\text{s} = 0{\cdot}01 \times 10^{-4} \text{ m}^2/\text{s}$$

Total drag on the prototype:

(*i*) *Analysis of model :*

Reynolds number, $$Re = \left(\frac{VL}{\nu}\right)_m = \frac{V_m L_m}{\nu_m}$$

$$= \frac{1{\cdot}5 \times 8}{0{\cdot}01 \times 10^{-4}} = 12 \times 10^6$$

Also $$C_f = \frac{0{\cdot}0735}{(Re)^{1/5}}$$

$\therefore$ $$C_{fm} = \frac{0{\cdot}0735}{(12 \times 10^6)^{1/5}} = 0{\cdot}00282$$

Frictional or skin resistance of the model is given by :

$$F_f = \frac{1}{2} C_f \rho A V^2$$

$\therefore$ $$(F_f)_m = (R_f)_m = \frac{1}{2} C_{fm} \rho_m A_m V_m^2 = \frac{1}{2} \times 0{\cdot}00282 \times \left(\frac{9810}{9{\cdot}81}\right) \times 5 \times 1{\cdot}5^2$$

$$= 15{\cdot}86 \, N$$

Also, $$R_m = (R_w)_m + (R_f)_m$$

$$\left[\begin{array}{l}\text{where } (R_w)_m = \text{wave resistance experienced by the model, and} \\ (R_f)_m = \text{frictional or skin resistance experienced by the model.}\end{array}\right]$$

$\therefore$ $$20 = (R_w)_m + 15{\cdot}86$$

or $$(R_w)_m = 20 - 15{\cdot}86 = 4{\cdot}14 \text{ N}$$

(*ii*) *Analysis of prototype:*

Since resistance to wave formation exists, condition will be dynamically similar if the *Froude's numbers are equal.*

i.e. $$\frac{V_p}{\sqrt{L_p g_p}} = \frac{V_m}{\sqrt{L_m g_m}}$$

or $$V_p = V_m \times \sqrt{\frac{L_p}{L_m}} = 1{\cdot}5 \times \sqrt{20} = 6{\cdot}708 \text{ m/s} \qquad (\because g_p = g_m)$$

Reynolds number, $$Re = \left(\frac{VL}{\nu}\right)_p = \frac{V_p L_p}{\nu_p}$$

$$= \frac{6{\cdot}708 \times 160}{0{\cdot}01 \times 10^{-4}} = 10{\cdot}73 \times 10^8 \qquad \left[\begin{array}{l}\because \dfrac{L_p}{L_m} = 20 \\ \therefore L_p = 8 \times 20 = 160 \text{ m}\end{array}\right]$$

$\therefore$ $$C_{fp} = \frac{0{\cdot}0735}{(10{\cdot}73 \times 10^8)^{1/5}} = 0{\cdot}001148$$

∴ Frictional or skin resistance of the prototype is given by:

$$(F_f)_p = (R_f)_p = \frac{1}{2} C_{fp} \rho_p A_p V_p^2 = \frac{1}{2} \times 0.001148 \times \frac{9810}{9.81} \times 2000 \times 6.708^2$$

$$= 51657 \text{ N} \qquad \left[\because \; A_p = A_m \times L_r^2 = 5 \times 20^2 = 2000 \text{ m}^2 \right]$$

Wave resistance R_w for the prototype can be evaluated by satisfying the following condition for dynamic similarity:

$$\left(\frac{R_w}{\rho L^2 V^2} \right)_p = \left(\frac{R_w}{\rho L^2 V^2} \right)_m$$

∴ $$(R_w)_p = \frac{\rho_p}{\rho_m} \times \left(\frac{L_p}{L_m} \right)^2 \times \frac{V_p^2}{V_m^2} \times (R_w)_m$$

But $\rho_p = \rho_m$ ...(Given)

and $\left(\frac{V_p}{V_m} \right)^2 = \frac{L_p}{L_m}$ ...[From the equivalent of Froude's number]

∴ $$(R_w)_p = 1 \times \left(\frac{L_p}{L_m} \right)^3 \times (R_w)_m = (20)^3 \times 4.14 = 33120 \text{ N}$$

Hence total drag on the prototype,

$$R_p = (R_w)_p + (R_f)_p = 33120 + 51657 = 84777 \text{ N} \quad \textbf{(Ans.)}$$

5. Euler model law,

In a fluid system where *pressure forces alone are the controlling forces in addition to the inertia force*, the dynamic similarity is obtained by equating the Euler number for both the model and its prototype. This is known as *Euler model law*. According to this law:

$$(Eu)_{\text{model}} = (Eu)_{\text{prototype}}$$

If, V_m = velocity of fluid in model,

p_m = pressure of fluid in model,

ρ_m = density of fluid in model,

and V_p, p_p, ρ_p are the corresponding values in prototype, then by substituting these values in eqn. (34), we get

$$\frac{V_m}{\sqrt{p_m/\rho_m}} = \frac{V_p}{\sqrt{p_p/\rho_p}}$$

when $\rho_m = \rho_p$, (*i.e.* same fluid in model and prototype) the above equation becomes

$$\frac{V_m}{\sqrt{p_m}} = \frac{V_p}{\sqrt{p_p}}$$

This law is applied in the following flow problems :

(*i*) Enclosed fluid system where the turbulence is fully developed so that viscous forces are negligible and also the forces of gravity and surface tensions are entirely absent;

(*ii*) Where the phenomenon of cavitation occurs.

Example . *In an aeroplane model of size $\frac{1}{10}$ of its prototype the pressure drop is $7{\cdot}5\ kN/m^2$. The model is tested in water. Find the corresponding pressure drop in the prototype.*

Take: Density of air $= 1{\cdot}24\ kg/m^3$; Density of water $= 1000\ kg/m^3$

Viscosity of air $= 0{\cdot}00018$ poise; Viscosity of water $= 0{\cdot}01$ poise.

Sol. Linear scale ratio, $L_r = 40$; Pressure drop in model, $(\Delta p)_m = 7{\cdot}5\ \text{kN/m}^2$;

Density of water, $\rho_m = 1000\ \text{kg/m}^3$; Viscosity of water, $\mu_m = 0{\cdot}01$ poise;

Density of air, $\rho_p = 1{\cdot}24\ \text{kg/m}^3$; Viscosity of air, $\mu_p = 0{\cdot}00018$ poise.

Pressure drop in the prototype, $(\Delta P)_p$:

Since in the problem pressure and viscous forces are involved, therefore, for dynamic similarity between the model and prototype, Euler's number and Reynolds number should be considered.

Making Reynolds number equal, we get

$$\left(\frac{\rho VL}{\mu}\right)_m = \left(\frac{\rho VL}{\mu}\right)_p$$

or
$$\frac{\rho_m V_m L_m}{\mu_m} = \frac{\rho_p V_p L_p}{\mu_p}$$

or
$$\frac{V_m}{V_p} = \frac{\rho_p}{\rho_m} \times \frac{L_p}{L_m} \times \frac{\mu_m}{\mu_p}$$

Substituting the values, we have

$$\frac{V_m}{V_p} = \frac{1{\cdot}24}{1000} \times 40 \times \frac{0{\cdot}01}{0{\cdot}00018} = 2{\cdot}755$$

Now making Euler's number equal, we get

$$\left(\frac{V}{\sqrt{p/\rho}}\right)_m = \left(\frac{V}{\sqrt{p/\rho}}\right)_p$$

or
$$\frac{V_m}{\sqrt{(\Delta p)_m/\rho_m}} = \frac{V_p}{\sqrt{(\Delta p)_p/\rho_p}}$$

or
$$\frac{V_m}{V_p} = \frac{\sqrt{(\Delta p)_m/\rho_m}}{\sqrt{(\Delta p)_p/\rho_p}} = \sqrt{\frac{(\Delta p)_m}{(\Delta p)_p}} \times \sqrt{\frac{\rho_p}{\rho_m}}$$

or
$$\sqrt{\frac{(\Delta p)_m}{(\Delta p)_p}} = \frac{V_m}{V_p} \times \sqrt{\frac{\rho_m}{\rho_p}} = 2{\cdot}755 \times \sqrt{\frac{1000}{1{\cdot}24}} = 78{\cdot}24$$

$\therefore$
$$\frac{(\Delta p)_m}{(\Delta p)_p} = (78{\cdot}24)^2 = 6121{\cdot}5$$

or
$$(\Delta p)_p = \frac{(\Delta p)_m}{6121{\cdot}5} = \frac{7{\cdot}5 \times 1000}{6121{\cdot}5}\ \text{N/m}^2 = 1{\cdot}225\ \text{N/m}^2$$

Hence pressure drop in the prototype $=$ **$1{\cdot}225\ \text{N/m}^2$ (Ans.)**

INTERNAL FLOWS

FLOW IN PIPES AND DUCTES

For the flow of real fluid in ducts, channels and pipes, many of the concepts developed before can be applied with only minor, but important, changes needed to account for viscous effects, boundary layers, separation and secondary flows.

The problem of fluid flow in pipelines-the prediction of flowrate through pipes of given characteristics, the calculation of energy conversions therein, and so forth-is encountered in many areas of engineering practice. The subject of pipe flow embraces only those problems in which pipes flow completely full; pipes that flow partially full, such as sewer lines and culverts.

The solution of practical pipe flow problems; results from application of the work-energy principle, the equation of continuity, and the principles and equations of fluid resistance. Resistance to flow in pipes is offered not only by long reaches of pipe but also by pipe fittings, such as bends and valves, which dissipate energy by producing relatively large-scale turbulence.

4.1- STEADY FLOW FUNDAMENTAL EQUATIONS

4.1.1- SHEARS STRESS AND HEAD LOSS

The effect of friction forces on the boundary surface inside a pipe, can be calculated by applying the linear impulse momentum principle, derived before in the form, $\sum_{abcd}\ddot{R}+\ddot{F}_{\Sigma}=\sum_{abcd}\left(\dot{m}\ddot{C}_2-\dot{m}\ddot{C}_1\right)$*, to a circular cylindrical pipe, in which the surface shear stress T_0 represent the effect of friction forces on the solid periphery (boundary) of passage opposing the direction of fluid motion, Fig. 3.1*

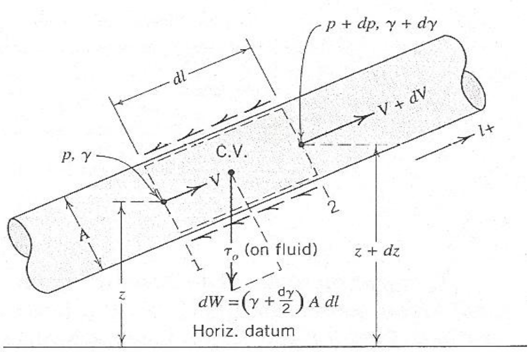


Fig. 3.1

Let us write the impulse momentum relation along the direction of this stream tube to the control volume bounded by sections 1, 2 and the stream tube boundary, as result:

$$pA - (p + dp)A - \tau_o P\, dl - \left(\gamma + \frac{d\gamma}{2}\right) A\, dl\, \frac{dz}{dl} = (V + dV)^2 A(\rho + d\rho) - V^2 A\rho$$

Where: ***S = πD****, (perimeter of the stream tube)*

m = ρAV = (ρ + dρ) A (V + dV) = const.

Introducing hydraulic radius;

For an established incompressible flow in a tube of constant cross section, τ_o is not a function of l, γ = const and d (1/ γ) =0, and we can write the last relation in the form:

$$\frac{dp}{\gamma} + d\left(\frac{V^2}{2g_n}\right) + dz = -\frac{\tau_o\, dl}{\gamma R_h}$$

$$d\left(\frac{p}{\gamma} + \frac{V^2}{2g_n} + z\right) = -\frac{\tau_o\, dl}{\gamma R_h}$$

Integrating this equation between 1 & 2, one has:

$$\left(\frac{p_1}{\gamma} + \frac{V_1^2}{2g_n} + z_1\right) - \left(\frac{p_2}{\gamma} + \frac{V_2^2}{2g_n} + z_2\right) = \frac{\tau_o(l_2 - l_1)}{\gamma R_h}$$

The left hand side of the above equation represents the drop of the energy line Δ (EL) between points 1&2. The energy line is also called the total head line, and the drop of the energy line is the head loss h_L

$$\left(\frac{p_1}{\gamma} + \frac{V_1^2}{2g_n} + z_1\right) - \left(\frac{p_2}{\gamma} + \frac{V_2^2}{2g_n} + z_2\right) = \Delta(\text{EL}) = \frac{\tau_o(l_2 - l_1)}{\gamma R_h}$$

And we can write:

$$\boxed{h_{L_{1-2}} = \frac{\tau_o(l_2 - l_1)}{\gamma R_h}}$$

The forgoing analysis may be similarly applied to any streamtube of the flow, e.g. streamtube with radius r and concentric with the axis of cylindrical pipe, fig.3.2. The frictional stress $\boldsymbol{\tau}$ *will be that exerted on the outer most fluid layer of the streamtube by the adjacent (more slowly moving) fluid. By analogy, we can write* $\boldsymbol{\tau}$ *for* $\boldsymbol{\tau_o}$, ***r/2*** *for* $\boldsymbol{R_h}$, $\boldsymbol{h_l}$ *for* $\boldsymbol{h_{l1-2}}$ *and* ***l*** *for* $\boldsymbol{l_1}$-$\boldsymbol{l_2}$*, from which:*

$$\boxed{\tau = \left(\frac{\gamma h_L}{2l}\right) r}$$

This shows that, in established pipe flow, the shear stress in the fluid varies linear with distance from the pipe centerline. As note both of the last two relations are valid for both laminar and turbulent flow in pipe as they have been developed without regard to the flow regime.

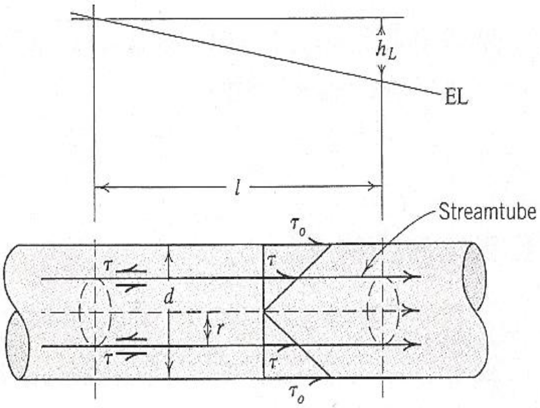


Fig. 3.2

Example.1

Water flows in a 0.9 m by 0.6 m rectangular conduit. The head lost in 60 m of this conduit is determined (experimentally) to be 10 m. Calculate the resistance stress exerted between fluid and conduit walls.

Solution

Since

$$h_{L_{1-2}} = \frac{\tau_o(l_2 - l_1)}{\gamma R_h}$$

$$R_h = A/P$$

and the data (given or obtained from Appendix 2) are

$$\gamma = 9.8\ \text{kN/m}^3 \qquad h_{L_{1-2}} = 10\ \text{m} \qquad l_2 - l_1 = 60\ \text{m}$$

$$A = 0.9 \times 0.6 = 0.54\ \text{m}^2 \qquad P = 2(0.9 + 0.6) = 3\ \text{m}$$

we have

$$\tau_o = \frac{10 \times 9.8 \times 10^3 \times 0.54/3}{60} = 0.29\ \text{kPa} \quad \bullet$$

If the results cited in the preceding problem are obtained for water flowing in a cylindrical pipe 0.6 m in diameter, what shear stress is to be expected (*a*) between fluid and pipe wall, and (*b*) in the fluid at a point 200 mm from the wall?

The additional relevant equation is

$$\tau = \left(\frac{\gamma h_L}{2l}\right) r$$

and

$$d = 0.6 \text{ m} \qquad r = 0.1 \text{ m}$$

See the previous problem for Eq. 7.36, γ, $h_{L_{1-2}} = h_L$, $l = l_2 - l_1$, R_h, etc.

Clearly,

$$R_h = d/4; \qquad \text{then}$$

$$\tau_o = \left(\frac{10 \times 9.8 \times 10^3}{60}\right) 0.15 = 0.25 \text{ kPa}$$

From the linear variation of τ with r (Eq. 7.37),

$$\tau = \tfrac{100}{300}(0.25 \times 10^3) = 83.3 \text{ Pa} = 0.083\,3 \text{ kPa}$$

4.1.2- General Energy Equation for Steady Incompressible Flow:

Considering the work energy equation; for the control volume which coincides with the walls of streamtube or, full-sized conduit, Fig, 3.3.

$$\frac{dQ}{dt} + \frac{dW}{dt} = \frac{dE}{dt}$$

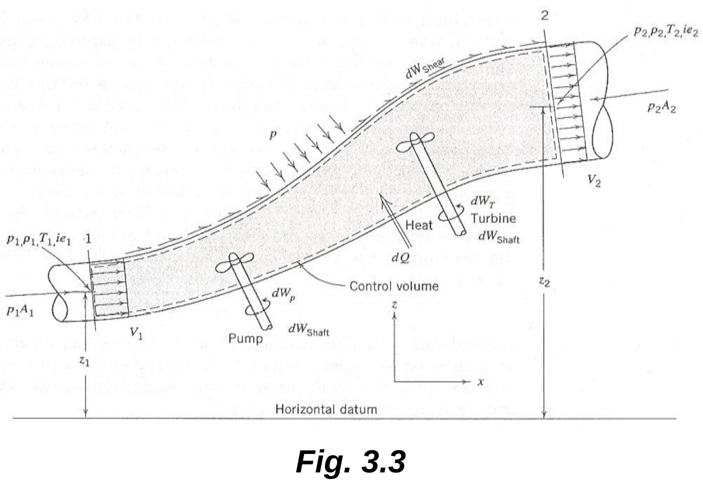


Fig. 3.3

Starting from the equation considering the unit mass of substance, which has the

form:

$\dot{m} = \rho_1 A_1 V_1 = \rho_2 A_2 V_2$ in this steady flow. Therefore,

$$\frac{dQ}{dt} = \dot{m} q_H \quad \text{or} \quad q_H = \frac{1}{\dot{m}} \frac{dQ}{dt} = \left[\frac{\text{N}\cdot\text{m}}{\text{kg}}\right] = \left[\frac{\text{joule}}{\text{kg}}\right] = \left[\frac{\text{ft}\cdot\text{lb}}{\text{slug}}\right]$$

Now dividing the equation by g and considering incompressible fluid; $\rho_1=\rho_2=\rho$, *noting that:*

$$\frac{dW_{\text{shaft}}}{dt} = Q\gamma(E_P - E_T) = \dot{m} g_n (E_P - E_T)$$

$$\frac{1}{\dot{m}} \frac{dW_{\text{shaft}}}{dt} = g_n E_P - g_n E_T = \left[\frac{\text{J}}{\text{kg}}\right] = \left[\frac{\text{ft}\cdot\text{lb}}{\text{slug}}\right]$$

Where: $\boldsymbol{E_T}$, *is the energy extracted by the turbine per unit weight of fluid flowing,* ***[J/N]*** $\boldsymbol{E_p}$, *is the energy added by the pump per unit weight of fluid flowing,* ***[J/N =m]***

$$\frac{dW_{\text{flow}}}{dt} = p_1 A_1 V_1 - p_2 A_2 V_2 \quad \text{or} \quad \frac{1}{\dot{m}} \frac{dW_{\text{flow}}}{dt} = \frac{p_1}{\rho_1} - \frac{p_2}{\rho_2} = \left[\frac{\text{J}}{\text{kg}}\right] = \left[\frac{\text{ft}\cdot\text{lb}}{\text{slug}}\right]$$

The total energy of the system is given by

$$E = \iiint_{System} i \cdot dm = \iiint_{System} (\text{kinetic} + \text{potential} + \text{internal energies}) \cdot dm$$

$$= \iiint_{System} \left(\frac{1}{2} V^2 + g_n z + ie\right) \cdot \rho \, d\mathcal{V}$$

$U_2 - U_1$, *change of specific internal energy of fluid. Then the work energy equation takes the form:*

$$\frac{dE}{dt} = \iint_{C.S._{out}} i \cdot (\rho \mathbf{v} \cdot d\mathbf{A}) + \iint_{C.S._{in}} i \cdot (\rho \mathbf{v} \cdot d\mathbf{A})$$

Which; is known as "first law of thermodynamics for a control volume". For which, when we neglect the effect of pump and turbine, it takes the form:

$$\frac{dE}{dt} = \iint_{C.S._{out}} \left(\frac{1}{2} V^2 + g_n z + ie\right)(\rho \mathbf{v} \cdot d\mathbf{A}) + \iint_{C.S._{in}} \left(\frac{1}{2} V^2 + g_n z + ie\right)(\rho \mathbf{v} \cdot d\mathbf{A})$$

The left hand side of this equation represent the head loss h_L*\-2 (as explained in section 8.10), the flow in both cases apply precisely the same conditions (incompressible flow in tubes). And we can write*

$$\frac{dE}{dt} = \left(\frac{1}{2}V^2 + g_n z + ie\right)_2 \cdot (\rho_2 V_2 A_2) - \left(\frac{1}{2}V^2 + g_n z + ie\right)_1 \cdot (\rho_1 V_1 A_1)$$

However, $\dot{m} = \rho_2 V_2 A_2 = \rho_1 V_1 A_1$; thus,

This equation offers proof that head loss is not a loss of total energy but rather a conversion of energy into heat, part of which leaves the fluid, the remainder serving to increase its internal energy.

$$\frac{1}{\dot{m}}\frac{dE}{dt} = \left(\frac{1}{2}V^2 + g_n z + ie\right)_2 - \left(\frac{1}{2}V^2 + g_n z + ie\right)_1$$

Here head loss is a permissible and useful concept because heat energy leaving the flow and energy converted into internal energy are seldom recoverable and are in effect lost from the useful total energy (pressure, velocity and potential energies).

$$q_H + (g_n E_P - g_n E_T) + \left(\frac{p_1}{\rho_1} - \frac{p_2}{\rho_2}\right) = \left(\frac{1}{2}V^2 + g_n z + ie\right)_2 - \left(\frac{1}{2}V^2 + g_n z + ie\right)_1$$

$$\left(\frac{p_1}{\gamma_1} + \frac{V_1^2}{2g_n} + z_1\right) + E_P = \left(\frac{p_2}{\gamma_2} + \frac{V_2^2}{2g_n} + z_2\right) + E_T + \frac{1}{g_n}(ie_2 - ie_1 - q_H)$$

This is riot generally true, since the useful Total of energies will include the internal energy. Introducing the last equation generates a reasonably (but not completely) general energy equation for steady-incompressible flow, in the form:

$$\left(\frac{p_1}{\gamma} + \frac{V_1^2}{2g_n} + z_1\right) + E_P = \left(\frac{p_2}{\gamma} + \frac{V_2^2}{2g_n} + z_2\right) + E_T + h_{L_{1-2}}$$

This equation is derived considering the mean streamline concept therefore; correction factors should be applied to allow for nonuniform distribution of fluid" parameters at each section; (velocity, pressure, density and temperatures).

Example3:

A flowrate of 1.42 m^3/s of water occurs in a streamtube, say a pipe of variable cross section, containing a pump, but no turbine. The pump delivers 300 kW to the flowing fluid. Measurements at two points, i.e., 1 and 2 along the streamtube, show that $A_1 = 0.4$ m^2, $A_2 = 0.2$ m^2, $z_1 = 9$ m, $z_2 = 24$ m, $p_1 = 138$ kPa, $p_2 = 69$ kPa. Calculate the head lost between sections 1 and 2.

Solution:

Taking $\gamma = 9.81\ \text{kN/m}^3$ and using Eqs. 4.4 and 5.8 produces

$$Q = AV,\ 1.42 = 0.4V_1 = 0.2V_2 \qquad V_1 = 3.55\ \text{m/s} \qquad V_2 = 7.1\ \text{m/s}$$

$$P = Q\gamma E_P \quad \text{and so} \quad E_P = (300 \times 10^3)/(9\,810 \times 1.42)$$
$$= 21.54\ \text{J/N} = 21.54\ \text{N·m/N} = 21.54\ \text{m}$$

Apply Eq. 7.46 with $E_T = 0$ to find (recall $g_n = 9.81\ \text{m/s}^2$)

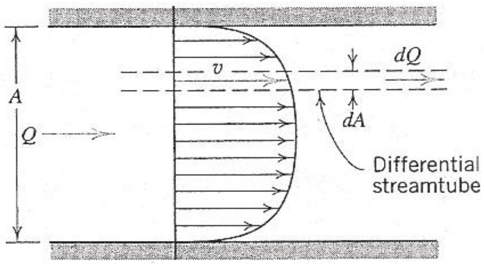
$$\left(\frac{p_1}{\gamma_1} + \frac{V_1^2}{2g_n} + z_1\right) + E_P = \left(\frac{p_2}{\gamma_2} + \frac{V_2^2}{2g_n} + z_2\right) + E_T + h_{L_{1-2}}$$

$$\left(\frac{138\,000}{9\,810} + \frac{(3.55)^2}{2g_n} + 9\right) + 21.54 = \left(\frac{69\,000}{9\,810} + \frac{(7.1)^2}{2g_n} + 24\right) + h_{L_{1-2}}$$

$$h_{L_{1-2}} = 11.7\ \text{m} \quad \bullet$$

4.1.3- Velocity Distribution and its Significance

Due to viscous effect (shear stress effect), we have non-uniform velocity distribution, characterized by reduced velocities near boundary surface, Fig. 3.4. In this case we can't consider the mean streamline concept, in which the values on the cross sectional area is considered constant and equal to the values on the mean line. We must count for the non-uniform velocity distribution by taking the integration over the cross section.



$$\text{Total kinetic energy (J/s or ft·lb/s)} = \frac{\rho}{2}\iint_A v^3\, dA$$

$$\text{Momentum flux (N or lb)} = \rho \iint_A v^2\, dA$$

For the volume flow rate:

$$Q = \int_A dQ = \int_A V dA \quad ; \quad \dot{m} = \int_A dm = \int_A \rho v dA$$

For total kinetics energy:

$$KE = \int_A \rho dQ \frac{v^2}{2} = \int_A \rho v dA \frac{v^2}{2} = \int_A \rho \frac{v^3}{2} dA = \frac{\gamma}{2} \int_A \frac{v^3}{g} dA$$

For momentum flux:

$$MF = \int_A \rho dQ v = \int_A \rho v dA v = \int_A \rho v^2 dA = \rho \int_A v^2 dA = \gamma \int_A \frac{v^2}{g} dA$$

However we can use the mean velocity V, and total flow rate Q for expressing total kinetic energy and momentum flux by introducing correction factor a, p (dimensionless) defined as:

$$\text{Total kinetic energy} = \alpha Q \gamma \frac{V^2}{2g_n} = Q\gamma \left(\alpha \frac{V^2}{2g_n} \right)$$

$$\text{Momentum flux} = \beta Q \rho V$$

For momentum flux and total kinetics energy

$$\boxed{\alpha = \frac{1}{V^2} \frac{\iint_A v^3 \, dA}{\iint_A v \, dA}} \quad \text{and} \quad \boxed{\beta = \frac{1}{V} \frac{\iint_A v^2 \, dA}{\iint_A v \, dA}}$$

*Where **α=β=1** for uniform flow, and **α >β>1** for non uniform velocity profiles.*

Notes:*For ideal incompressible fluid we prove that -.the quantity **(z+p\γ)** is constant over flow cross section normal to the streamlines when they are straight and parallel, which is often called the hydrostatic pressure distribution as it is the same relation for a fluid at rest. For the established flow of a real fluid in prismatic passage (where the shear stresses perpendicular to the direction of motion may be neglected, which is the case of almost engineering problems except for very low Reynolds number and low velocity), the hydrostatic pressure distribution is not affected by viscosity, but only by the streamline curvature.*

Therefore the quantity any flow cross section, and the Energy line concept is no more valid, and must be reexamined.

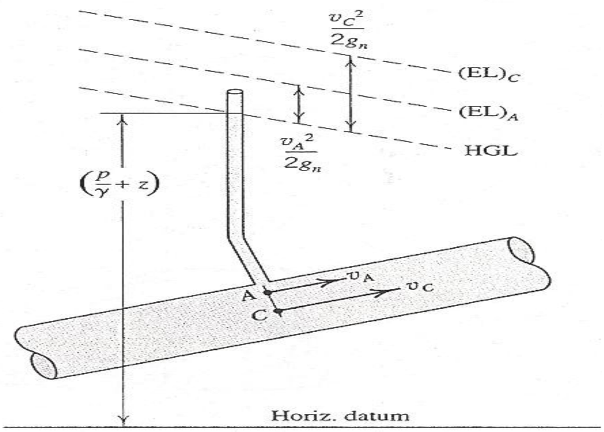
Taking A and C as typical streamlines, Fig.4.5, it is noted that each streamline is associated with a different energy line, i.e. the flow along different streamlines possesses different, amount of total energy.

However, for flow-in passage, as pipe, duct, or open channel, properties of individual streamlines are seldom of interested the whole flow is characterized by a single effective energy line a distance $\frac{\alpha V^2}{2g}$ *above the hydraulic gradient line.*

Another consequence (effect) of non-uniform velocity distribution can be shown by considering a flow through a short constriction, Fig. 3.6. From continuity $V_2 > V1$

Hydrostatic relation is valid $\left(\frac{p}{\gamma}+z\right)_1 = const$; $\left(\frac{p}{\gamma}+z\right)_2 = const$

Frictional effect of constriction is relatively small, and as the acceleration is an efficient process, the velocity profile at sec is flatter than that at sec 1, then accordingly $\alpha_2 \langle \alpha_1$ *(correction factor)*

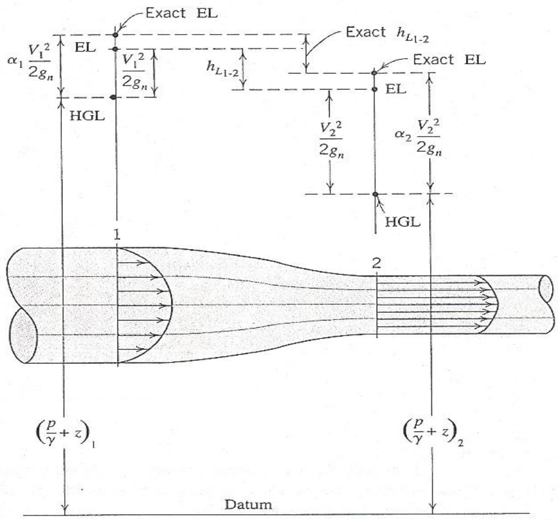


And now let us calculate exact and conventional head loss, exact $h_{L\ 1\_2}$ *and* $h_{L\ 1\_2}$ *(head loss is given by drop in energy line)*

$$\text{Exact } h_{L_{1-2}} = \left(\alpha_1 \frac{V_1^2}{2g_n} + \frac{p_1}{\gamma} + z_1\right) - \left(\alpha_2 \frac{V_2^2}{2g_n} + \frac{p_2}{\gamma} + z_2\right)$$

And $h_{L1\text{-}2}$ *is given by ignoring* α,

$$h_{L_{1-2}} = \left(\frac{V_1^2}{2g_n} + \frac{p_1}{\gamma} + z_1\right) - \left(\frac{V_2^2}{2g_n} + \frac{p_2}{\gamma} + z_2\right)$$



comparing both equations gives

$$h_{L_{1-2}} = \text{Exact } h_{L_{1-2}} + (\alpha_2 - 1)\frac{V_2^2}{2g_n} - (\alpha_1 - 1)\frac{V_1^2}{2g_n}$$

as $V_2 > V_1$ and $\alpha_2 \langle \alpha_1$, some compensating features exist, and therefore Exact $h_{L12} \approx h_{L1\text{-}2}$.

- *Only one value of change of (+ z), between section 1&2*

The work-energy equation for incompressible fluid motion in pipes is:

$$Z_1 + \frac{p_1}{\gamma} + \alpha_1 \frac{v_1^2}{2g} = Z_2 + \frac{p_2}{\gamma} + \alpha_2 \frac{v_2^2}{2g} + h_L \qquad (3)$$

However, in most problems of pipe flow, α term may be omitted for several reasons:

(1) Most engineering pipe flow problems involve turbulent flow in which α is only slightly more than unity.

(2) In laminar flow where α is large, velocity heads are usually negligible when compared to the other terms.

(3) The velocity heads in most pipe flows are usually so small compared to other terms that inclusion of α has little effect on the final result.

(4) Engineering answers are not usually required to an accuracy which would justify the inclusion of α in the equation.

Example 2

Assuming Fig. 7.24 to represent a parabolic velocity profile in a passage bounded by two infinite planes of spacing $2R$ and maximum velocity v_c, calculate q, α, and β.

Solution:

The necessary equations are Eqs. 4.8 and 7.51, namely,

$$q = VA = V(2R) = \iint_A v\, dA$$

$$\alpha = \frac{1}{V^2}\frac{\iint_A v^3\, dA}{\iint_A v\, dA} \quad \text{and} \quad \beta = \frac{1}{V}\frac{\iint_A v^2\, dA}{\iint_A v\, dA}$$

Taking r as the distance from centerline of the passage to any local velocity, v, and element of area dA, $dA = dr$. The equation of the parabola is $v = v_c(1 - r^2/R^2)$.

$$q = 2\int_0^R v_c\left(1 - \frac{r^2}{R^2}\right)dr = \frac{2}{3}(2Rv_c)$$

Since q also equals $2RV$, $V = 2v_c/3$.

$$\alpha = \frac{2\int_0^R v_c^3\left(1 - \frac{r^2}{R^2}\right)^3 \mathrm{dr}}{\left(\frac{2v_c}{3}\right)^2 \frac{2}{3}(2Rv_c)} = \frac{54}{35} \doteq 1.54$$

$$\beta = \frac{2\int_0^R v_c^2\left(1 - \frac{r^2}{R^2}\right)^2 dr}{\left(\frac{2v_c}{3}\right) \frac{2}{3}(2Rv_c)} = \frac{6}{5} = 1.20$$

The meaning of these figures is that the exact velocity head is more than 54% greater than $V^2/2g_n$, and the exact momentum flux 20% greater than $Q\rho V$. Differences of this magnitude warn the engineer that α and β should be considered when applying the energy and momentum equations to one-dimensional laminar flow problems—unless their effects can be shown to have negligible consequence in the results desired.

4.1.4- NAVIER-STOKES EQUATIONS

We would like to derive the differential form of the momentum conservation law for incompressible flows. We shall first choose an infinitesimally small control volume $\Delta V = \Delta x \cdot \Delta y \cdot \Delta z$ *around the point (x, y, and z) and then shrink this volume to a point by allowing ΔV Recall from the Newton's*

$$\sum \vec{F}_B + \sum \vec{F}_S = m\vec{a}$$

second law of motion:

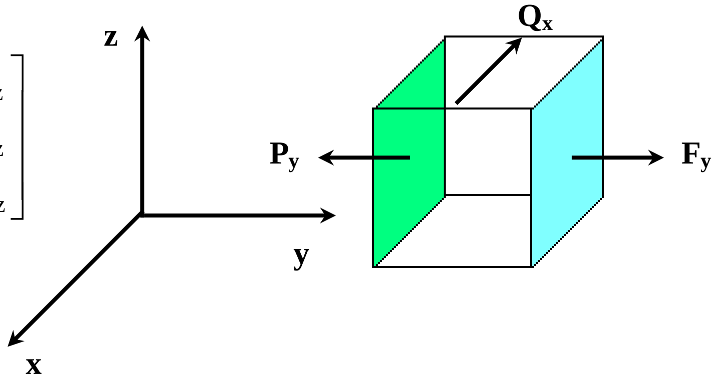
Where, the left hand side represents the sum of the external forces and the right hand side represents the inertia forces. Since $\vec{a} = \frac{D\vec{V}}{Dt}$ *represent the total acceleration at a point,*

Let us review some facts about stresses. The stress tensor that you were taught in the strength of materials and represented by the matrix below has 9 components.

Note that "σ" represents a normal stress and "τ" represents a shear stress. Since stress is defined as a force over an area, each stress component has two subscripts. For example, $\sigma_{xx} = \frac{F_x}{A_x}$ *represents a force in the x-direction that is also acting over an area that is in the x-direction.*

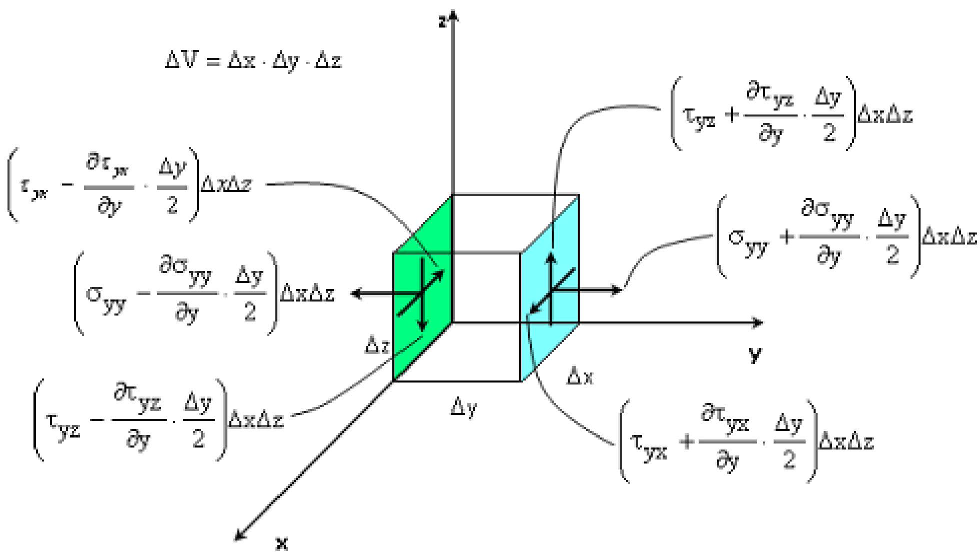
Now, an area may be considered positive or negative, depending on whether the outward normal to the area points in the positive or negative coordinate direction. For example, in the diagram below, we have chosen a positive and a negative y-area.

$$\begin{bmatrix} \sigma_{xx} & \tau_{xy} & \tau_{xz} \\ \tau_{yx} & \sigma_{yy} & \tau_{yz} \\ \tau_{zx} & \tau_{zy} & \sigma_{zz} \end{bmatrix}$$



Thus, the stress due to the force F_y shown on the positive y-face will represent a positive stress. Similarly, a positive stress on the negative y-face must be due to a force pointing in the negative y-direction, P_y. We may be able to represent shear stresses the same way. Question: Can you identify the shear stress due to the force Q_x shown in the above diagram? What will its subscript be and will it yield a positive or negative shear stress?

In the next set of figures we assume that the stress tensor is known at the center of a small control volume $\Delta V = \Delta x \cdot \Delta y \cdot \Delta z$ which is centered at (x, y, z). To eliminate the stress values on the surfaces of the control volume we apply <u>the Taylor series expansions</u> as before. We assume the stresses are all positive (i.e., forces act on positive faces in positive direction, or on negative faces in negative direction.). One such typical arrangement is shown on two y-faces below.



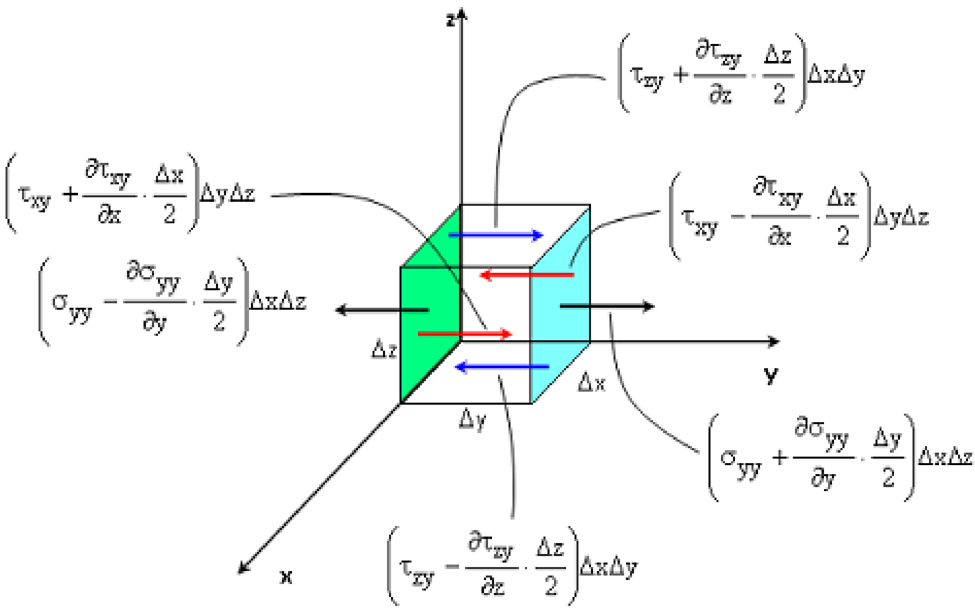
Let us examine the term $\left(\sigma_{yy} - \frac{\partial \sigma_{yy}}{\partial y} \cdot \frac{\Delta y}{2}\right) \Delta x \Delta z$*.The origin of* $\sigma_{yy}(x, y, z)$ *is at the center of the volume. However, since we would like to obtain the value of the stress on the "-y"-face, we must expand* $\sigma_{yy}(x, y - \frac{\Delta y}{2}, z)$ *using* <u>*Taylor series*</u> *about the* $\sigma_{yy}(x, y, z)$ *and its y-derivatives at (x,y,z):*

$$\sigma_{yy}(x, y - \frac{\Delta y}{2}, z, t) = \sigma_{yy}(x, y, z, t) - \frac{\partial \sigma_{yy}}{\partial y}(x, y, z, t) \cdot \frac{\Delta y}{2} + \frac{1}{2} \frac{\partial^2 \sigma_{yy}}{\partial y^2}(x, y, z, t) \cdot (\frac{\Delta y}{2})^2 - \ldots$$

But this provides only the shear value, the corresponding force in the "-y"-direction is given by $\left(\sigma_{yy} - \frac{\partial \sigma_{yy}}{\partial y} \cdot \frac{\Delta y}{2}\right) \Delta x \Delta z$ *as a first order approximation. Similarily the other stress terms are written.*

<u>*Derivation of Navier's Equations*</u>

Now let us focus on only the forces that act in the y-direction (instead of those that act on y-faces). Once this result is derived we may be able to write similar expressions for the x- and z-directions. For this purpose, use the following sketch:



$$\Delta F_{Sy} = \left(\sigma_{yy} + \frac{\partial \sigma_{yy}}{\partial y} \cdot \frac{\Delta y}{2} \right) \Delta x \Delta z - \left(\sigma_{yy} - \frac{\partial \sigma_{yy}}{\partial y} \cdot \frac{\Delta y}{2} \right) \Delta x \Delta z$$

$$+ \left(\tau_{xy} + \frac{\partial \tau_{xy}}{\partial x} \cdot \frac{\Delta x}{2} \right) \Delta y \Delta z - \left(\tau_{xy} - \frac{\partial \tau_{xy}}{\partial x} \cdot \frac{\Delta x}{2} \right) \Delta y \Delta z$$

$$+ \left(\tau_{zy} + \frac{\partial \tau_{zy}}{\partial z} \cdot \frac{\Delta z}{2} \right) \Delta x \Delta y - \left(\tau_{zy} - \frac{\partial \tau_{zy}}{\partial z} \cdot \frac{\Delta z}{2} \right) \Delta x \Delta y$$

$$= 2 \cdot \frac{\partial \sigma_{yy}}{\partial y} \cdot \frac{\Delta y}{2} \cdot \Delta x \Delta z + 2 \cdot \frac{\partial \tau_{xy}}{\partial x} \cdot \frac{\Delta x}{2} \cdot \Delta y \Delta z + 2 \cdot \frac{\partial \tau_{zy}}{\partial z} \cdot \frac{\Delta y}{2} \cdot \Delta x \Delta z$$

$$\sum F_{Sy} + \sum F_{By} = ma_y$$

Note that; the above shown are the only possible surface forces in the y-direction on the control volume. Therefore:

$$= \left(\frac{\partial \tau_{xy}}{\partial x} + \frac{\partial \sigma_{yy}}{\partial y} + \frac{\partial \tau_{zy}}{\partial z} \right) \Delta x \Delta y \Delta z$$

Body Force along $y \Rightarrow \Delta F_{By} = \rho B_y \cdot \Delta x \Delta y \Delta z$

Now we are ready to write the y-component of the momentum equation:

Since the mass of the small control volume chosen is $\rho \Delta V = \rho \Delta x \Delta y \Delta z$ *, and* $a_y = \frac{Dv}{Dt} = \frac{\partial v}{\partial t} + u \frac{\partial v}{\partial x} + v \frac{\partial v}{\partial y} + w \frac{\partial v}{\partial z}$ *, thus,*

$$\left(\frac{\partial \tau_{xy}}{\partial x} + \frac{\partial \sigma_{yy}}{\partial y} + \frac{\partial \tau_{zy}}{\partial z} \right) \Delta x \Delta y \Delta z + \rho B_y \Delta x \Delta y \Delta z =$$

$$\rho \Delta x \Delta y \Delta z \left(\frac{\partial v}{\partial t} + u \frac{\partial v}{\partial x} + v \frac{\partial v}{\partial y} + w \frac{\partial v}{\partial z} \right)$$

Cancelling ΔV from both sides we get:

$$\left(\frac{\partial \tau_{xy}}{\partial x} + \frac{\partial \sigma_{yy}}{\partial y} + \frac{\partial \tau_{zy}}{\partial z} \right) + \rho B_y = \rho \left(\frac{\partial v}{\partial t} + u \frac{\partial v}{\partial x} + v \frac{\partial v}{\partial y} + w \frac{\partial v}{\partial z} \right)$$

The first three terms on the left hand side represent surface forces per unit volume and the fourth one is the body force per unit volume. Similiarly, the right hand side terms collectively represent the inertia force per unit volume.

Let us examine the term $\frac{\partial \tau_{xy}}{\partial x}$ *once again. This term is the surface force on the control volume that arises due to the net shear force between two "x" surfaces but acting along the y-direction. Likewise the second and third terms are due to the normal force and shear force between two y-surfaces and two z-surfaces respectively.*

Note there is a pattern which emerges from the summation of all the y-forces on the control volume. The stress terms' second subscripts are all y, representing the direction of the force, but the first subscripts that represent which areas the forces act on, change from x to z for each successive term. Similarly the derivatives change also from x to z for each term. The vector component is represented by y, and the velocity component is accordingly, v. Likewise the x-and the z-equations may also be written rather than derived. Finally all three equations are represented by:

$$\left(\frac{\partial \sigma_{xx}}{\partial x}+\frac{\partial \tau_{yx}}{\partial y}+\frac{\partial \tau_{zx}}{\partial z}\right)+\rho B_x=\rho\left(\frac{\partial u}{\partial t}+u\frac{\partial u}{\partial x}+v\frac{\partial u}{\partial y}+w\frac{\partial u}{\partial z}\right)$$

$$\left(\frac{\partial \tau_{xy}}{\partial x}+\frac{\partial \sigma_{yy}}{\partial y}+\frac{\partial \tau_{zy}}{\partial z}\right)+\rho B_y=\rho\left(\frac{\partial v}{\partial t}+u\frac{\partial v}{\partial x}+v\frac{\partial v}{\partial y}+w\frac{\partial v}{\partial z}\right)$$

$$\left(\frac{\partial \tau_{xz}}{\partial x}+\frac{\partial \tau_{yz}}{\partial y}+\frac{\partial \sigma_{zz}}{\partial z}\right)+\rho B_z=\rho\left(\frac{\partial w}{\partial t}+u\frac{\partial w}{\partial x}+v\frac{\partial w}{\partial y}+w\frac{\partial w}{\partial z}\right)$$

The above three equations are collectively called the Navier's equations, named after their originator. Note that these equations have 4 independent variables (x, y, z, and t) but 12 dependent variables (u, v, w, and the stress components). We shall assume that the body force component (which is usually due to gravity in mechanical engineering problems) are known. Therefore, the Navier's equations are not solvable since there are 9 more unknowns in these equations than the number of equations.

To make these equations solvable, Stokes proposed a set of constitutive relations given below. These relations, together with the continuity equation derived before make the Navier's equations solvable. The Stoke's relations in Cartesian Coordinates are:

$$\tau_{xy} = \tau_{yx} = \mu\left(\frac{\partial v}{\partial x} + \frac{\partial u}{\partial y}\right)$$

$$\tau_{yz} = \tau_{zy} = \mu\left(\frac{\partial w}{\partial y} + \frac{\partial v}{\partial z}\right)$$

$$\tau_{zx} = \tau_{xz} = \mu\left(\frac{\partial u}{\partial z} + \frac{\partial w}{\partial x}\right)$$

$$\sigma_{xx} = -p - \frac{2}{3}\mu\nabla\cdot\vec{V} + 2\mu\frac{\partial u}{\partial x}$$

$$\sigma_{yy} = -p - \frac{2}{3}\mu\nabla\cdot\vec{V} + 2\mu\frac{\partial v}{\partial y}$$

$$\sigma_{zz} = -p - \frac{2}{3}\mu\nabla\cdot\vec{V} + 2\mu\frac{\partial w}{\partial z}$$

In the above relations, μ is the dynamic viscosity and p is the thermodynamic pressure. Also note that since the stress tensor is symmetric (i.e., $\tau_{xy}=\tau_{yx}$, etc), the first three relations are written in a more compact form rather than using 6 lines.

Since our focus in this class is incompressible flows, we may be able to set $\nabla\cdot\vec{V} = 0$ in the above relations by the use of the continuity equation for incompressible flows. If we substitute the above equation into the Navier's equations and repeatedly use the continuity equation for incompressible flows (this work is left out for you to try as an exercise), we obtain the simplified form of the Navier's equations as:

$$\rho\left(\frac{\partial u}{\partial t} + u\frac{\partial u}{\partial x} + v\frac{\partial u}{\partial y} + w\frac{\partial u}{\partial z}\right) = \rho B_x - \frac{\partial p}{\partial x} + \mu\left(\frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2} + \frac{\partial^2 u}{\partial z^2}\right)$$

$$\rho\left(\frac{\partial v}{\partial t} + u\frac{\partial v}{\partial x} + v\frac{\partial v}{\partial y} + w\frac{\partial v}{\partial z}\right) = \rho B_y - \frac{\partial p}{\partial y} + \mu\left(\frac{\partial^2 v}{\partial x^2} + \frac{\partial^2 v}{\partial y^2} + \frac{\partial^2 v}{\partial z^2}\right)$$

$$\rho\left(\frac{\partial w}{\partial t} + u\frac{\partial w}{\partial x} + v\frac{\partial w}{\partial y} + w\frac{\partial w}{\partial z}\right) = \rho B_z - \frac{\partial p}{\partial z} + \mu\left(\frac{\partial^2 w}{\partial x^2} + \frac{\partial^2 w}{\partial y^2} + \frac{\partial^2 w}{\partial z^2}\right)$$

In the above equations, popularly called the Navier-Stokes equations, we have only 4 dependent variables (u, v, w, and p), once again assuming B_x, B_y, and B_z to be known. Thus adding the continuity equation to this set:

$$\nabla\cdot\vec{V} = \frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} + \frac{\partial w}{\partial z} = 0$$

We now have a set of 4 equations in 4 dependent variables, or unknowns. Therefore, the fluid dynamic problems for incompressible flows are now clearly seen to be solvable.

Or, simply:

$$\nabla \cdot \vec{V} = 0 \qquad \text{———} \qquad \textbf{(A)}$$

$$\rho \frac{D\vec{V}}{Dt} = \rho\vec{B} - \nabla p + \mu\nabla^2\vec{V} \qquad \text{———} \qquad \textbf{(B)}$$

Note that the first equation (A) is a scalar equation (Since $\nabla \cdot \vec{V} = \frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} + \frac{\partial w}{\partial z}$ *is a scalar). However, the second equation, (B), hides 3 equations (one each for the x, y, and z components) into a single equation. Thus we still have 4 equations in 4 unknowns.*

We have already discussed the physical significance of the equation (A). Equation (B) is the vector form of the Newton's second law of motion (except each term has been divided by ΔV during the derivation). Thus the left hand side represents inertia force per unit volume, while each of the three terms in the right hand side represents a type of external force per unit volume (body force, pressure force, and viscous force respectivily).

4.1.5- DARCY-WEISBACH EQUATION:

*Early experiments (circa 1850) on the flow of water in long, straight, cylindrical pipes (Fig.) indicated that head loss varied (approximately) directly with velocity head and pipe length, and inversely with pipe diameter. Using a dimensionless coefficient of proportionality **f**, called the **friction factor,** which known by*

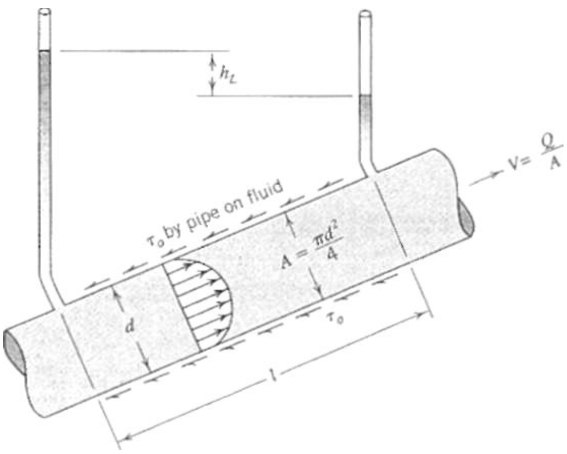


Fig. 4.6

$$h_L = f\,\frac{l}{d}\frac{v^2}{2g}$$

*The friction factor **f** depended primarily on pipe roughness in addition to velocity and pipe diameter; more recently it was observed that the friction factor **f** depended on the viscosity of the fluid flowing. From the relation between shear stress and head loss derived before;*

$$h_L = \frac{\tau_o l}{\gamma R_h}$$

Where R_h= hydraulic radius or hydraulic depth:

$$R_h = \frac{A}{P} = \frac{\pi}{4}\frac{d^2}{\pi d} = \frac{d}{4} = \frac{R}{2}$$

Equations 4.2 and 3.36 may now be combined to give a basic relation between frictional stress, τ, and friction factor, this is:

$$\tau_o = \frac{f\rho V^2}{8}$$

*In this fundamental equation relating wall shear to friction factor, density, and mean velocity, it is apparent that, with dimensionless, Vt_0/p must have the dimensions of velocity; this is known as the **friction velocity, v_*,** which is related to the friction factor and the mean velocity by:*

$$v_* = \sqrt{\frac{\tau_o}{\rho}} = V\sqrt{\frac{f}{8}} \qquad (4.4)$$

However, the physical meaning of the friction velocity is not revealed by this algebraic definition; since it is a velocity which embodies only wall shear and fluid density, it is defined by the same equation whatever the flow regime (laminar or turbulent) or whatever the boundary texture (rough or smooth). For this reason it is a useful generalization that finds wide application in further developments.

Example

Water flows in a 150 mm diameter pipeline at a mean velocity of 4.5 m/s. The head lost in 30 m of this pipe is measured experimentally and found to be 5.33 m. Calculate the friction velocity in the pipe.

SOLUTION:

We will use Eq. 9.4 to calculate the friction velocity but first we must find the Darcy-Weisbach friction factor f. We resort to Eq. 9.2 because all the terms in that equation are known except f.

$$h_L = f \frac{l}{d} \frac{V^2}{2g_n} \tag{9.2}$$

Solving Eq. 9.2 for f gives

$$f = \frac{2g_n}{V^2} \frac{d}{l} h_L = \frac{2 \times 9.81}{(4.5 \text{ m/s})^2} \frac{0.150 \text{ m}}{30 \text{ m}} 5.33 \text{ m} = 0.026$$

Now, bringing in Eq. 9.4,

$$v_* = V \sqrt{\frac{f}{8}} = 4.5 \text{ m/s} \sqrt{\frac{0.026}{8}} = 0.26 \text{ m/s} \tag{9.4}$$

4.2- LAMINAR FLOW

Analysis of laminar flow in a pipeline (Fig. 4.2) may be begun with the following established facts:

(1) symmetrical distribution of shear stress and velocity.

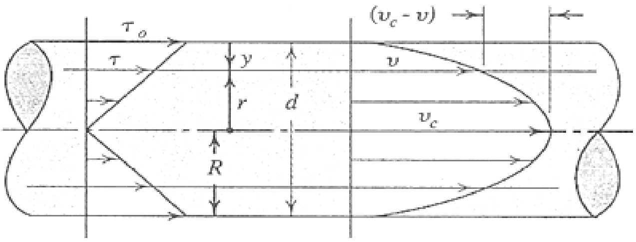
(2) maximum velocity at the center of the pipe and no velocity at the wall (the no-slip condition).

(3) linear shear stress distribution in the fluid given by the following Eqn (from application of the impulse-momentum principle.

$$\tau = \left(\frac{\gamma h_L}{2l}\right) r$$

(4) The shear stress in the laminar fluid flow is proportional to the velocity gradient as following:

$$= \mu \frac{dv}{dy}$$



In Fig. 4.2 it is clear that r / R - y and dr = - dy; thus, equating the expressions for r yields

$$\tau = \left(\frac{\gamma h_L}{2l}\right) r = \mu \frac{dv}{dy} = -\mu \frac{dv}{dr}$$

At the wall ***r=R, y=0*** *and* $\boldsymbol{\tau = \tau_o}$, *then* $\tau_o = \frac{\gamma h_L}{2l} R$, *from which,*

$\frac{\tau_o}{R} = \frac{\gamma h_L}{2l}$, *then,* $\frac{dv}{dr} = -\frac{1}{\mu}\tau = -\frac{1}{\mu}\left(\frac{\gamma h_L}{2l}\right) r = -\frac{1}{\mu}\frac{\tau_o}{R} r = -\frac{\tau_o}{\mu R} r$

Integrating this relation gives:

$$v = -\frac{\tau_o}{\mu R}\frac{r^2}{2} + C$$

From the no slip condition at r=R, V=0, then $0 = -\frac{\tau_o}{\mu R}\frac{R^2}{2} + C$, *then*

$C = \frac{\tau_o}{2}\frac{R^2}{\mu R}$, *from which the velocity profile is found to be parabolic in the form:*

$$v = \frac{\tau_o}{2\mu R}(R^2 - r^2) \qquad (9.5)$$

For *r=0 and* $v = V_c = \frac{\tau_o R^2}{2\mu R}$, *from which* $v = V_c\left(1 - \frac{r^2}{R^2}\right)$

Noting that $v_*^2 = \frac{\tau_o}{\rho}$, *so , the previous equation can be written as:*

$$v = \frac{v_*^2}{2\nu R}(R^2 - r^2) \qquad \text{or} \qquad \frac{v}{v_*} = \frac{v_*}{2\nu R}(R^2 - r^2) \qquad (9.6)$$

or with $r^2 = (R - y)^2$,

$$\frac{v}{v_*} = \frac{v_*}{\nu}\left(y - \frac{y^2}{2R}\right) \qquad (9.6)$$

$$\frac{v}{v_*} = \nu\left(y - \frac{y^2}{2R}\right)$$

Thus, the velocity profile can be expressed in terms of distance from the wall and is seen to be a characteristic "velocity" of the laminar profile.

It is interesting to note that near the pipe wall, the term $y^2/2R$ becomes negligible compared to y and the velocity profile near the wall is essentially linear. In dimensionless form, Eqn. becomes:

$$\frac{v}{v_*} = \frac{v_* y}{\nu}, \text{ where } y \langle\langle\langle R$$

We will approach development of a head loss equation by finding the flowrate Q by integrating the velocity profile given by Eqn.

$$Q = \int_0^R v(2\pi dr) = \frac{\pi\tau_o}{\mu R}\int_0^R \left(R^2 - r^2\right)dr = \frac{\pi\tau_o R^3}{4\mu}$$ *, substituting the relation for the wall shear stress* $\tau_o = \frac{\gamma h_L R}{2l}$*, then:*

$$Q = \frac{\pi R^4 \gamma h_L}{8\mu l} = \frac{\pi\, d^4 \gamma h_L}{128\mu l} \tag{9.8}$$

This is Hagen-Poiseuille law. Since $Q=\pi R^2 V$, the last equation takes the form:

$$V = \frac{\gamma R^2 h_L}{8\mu l} = \frac{\gamma\, d^2 h_L}{32\mu l}$$

From which head loss h_L equal to:

$$\boxed{h_L = \frac{32\mu l V}{\gamma\, d^2}} \tag{9.9}$$

Equation 4.8 shows that in laminar flow the flowrate, Q_y which will occur in a circular pipe, varies directly with the head loss and with the fourth power of the diameter but inversely with the length of pipe and viscosity of the fluid flowing. These facts of laminar flow were established experimentally, independently, and almost simultaneously by Hagen (1839) and Poiseuille (1840).

Equation 4.9 shows that, in a laminar flow, head loss varies with the first power of the velocity. Equating the Darcy-Weisbach equation 4.2 for head loss to Eqn. 4.9 yields an expression for the friction factor; it is

$$\boxed{f = \frac{64\mu}{Vd\rho} = \frac{64}{\mathbf{R}}} \tag{9.10}$$

Thus, in laminar flow the friction factor depends only on the Reynolds number.

Example

A fluid flows from a large pressurized tank through a 100 mm long, 4 mm diameter tube. In a 600 sec time period, 1 300 cm^3 of fluid are collected in a measuring cup. If the head loss in the tube is 1 m, calculate the kinematic viscosity ν. Check to verify that the flow is laminar.

SOLUTION:

First, we will calculate the flowrate by the fundamental relationship Q = volume/time. The volume is 1 300 cm^3/(10^6 cm^3/m^3) = 13 × 10^{-4} m^3. The time is 600 s.

$$Q = \frac{\text{Volume}}{\text{Time}} = \frac{13 \times 10^{-4}}{600} = 2.17 \times 10^{-6}\ \text{m}^3/s$$

We will use a modified version of Eq. 9.8 to find ν.

$$Q = \frac{\pi\, d^4 \gamma h_L}{128\mu l} = \frac{\pi\, d^4(\rho g_n)h_L}{128\mu l} = \frac{\pi\, d^4 g_n h_L}{128(\mu/\rho)l} = \frac{\pi\, d^4 g_n h_L}{128\nu l} \tag{9.8}$$

Now, solving the above equation for ν,

$$\nu = \frac{\pi\, d^4 g_n h_L}{128 Q l} = \frac{3.14 \times (4/1\,000)^4 \times 9.81 \times 1}{128 \times (2.17 \times 10^{-6}) \times (100/1\,000)} = 0.000\,28\ \text{m}^2/s \ \bullet$$

To calculate the Reynolds number, we need the average velocity V.

$$V = \frac{Q}{A} = \frac{2.17 \times 10^{-6}}{(\pi/4) \times (4/1\,000)^2} = 0.17\ \text{m/s}$$

Using Eq. 8.11 to obtain Reynolds number, we get

$$\mathbf{R} = \frac{Vd}{\nu} = \frac{0.17 \times (4/1\,000)}{0.000\,28} = 2.4 \tag{8.11}$$

Since $\mathbf{R}$ = 2.4 is well below 2 100, we have a strongly laminar flow and the analysis is correct. ●

4.3- TURBULENT FLOW

4.3.1– SMOOTH PIPES

Pipe flow with friction effects *has been seen to be a viscosity-inertia phenomenon and, thus, to be characterized by a Reynolds number. For smooth pipes, the discussion about flow past solid boundaries is relevant and strongly suggests the existence of a viscous sublayer near the pipe walls.*

The total shear stress can be written as:

$$\tau = \left(\mu \frac{dv}{dy} - \rho \overline{v_x v_y}\right) \qquad (9.11)$$

Where, μ dv/dy is the viscous stress and $-\rho\overline{v_x v_y}$ *is the turbulent (Reynolds) stress.*

It has already been shown that the total stresses are a linear function of radius r in a pipe. Experiments show that over most of the flow, the turbulent stress dominates, but the maximum stress equals the viscous stress at the wall, where the turbulent shear stress is zero.

The analytical treatment is begun by assuming that the viscous stress in Eqn. 9.11 is negligible over most of the flow, employing the Prandtl relationship $\tau = -\rho\overline{v_x v_y} = \rho l^2\left(\frac{dv}{dy}\right)^2$, *for the turbulent shear stress, and equating this to the linear total shear stress relation* $\tau = \left(\frac{\gamma h_L}{2l}\right) r$, *for pipes taking r = R - y and dr = - dy),then:*

$$\tau_o\left(1 - \frac{y}{R}\right) = \rho l^2 (dv/dy)^2$$

where y is measured from the pipe wall. Now in Section 7.2 it was argued that near the wall (y<<< R), l = Ky. However, here on the centerline (y = R), τ = 0, so either l= 0 or dv/dy = 0 there (or both equal zero). Thus, some insight is needed to proceed further; it comes from experiment. Nikuradse's systematic and comprehensive measurements of velocity profiles, in the form:

$$\boxed{\frac{v_c - v}{v_*} = -2.5 \ln \frac{y}{R}} \quad \text{(All pipes)}$$

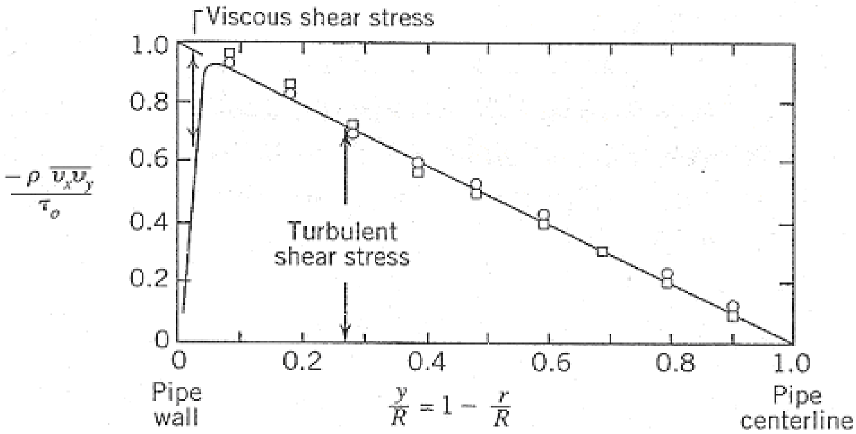


Fig.4. Typical variation of shear stress in a turbulent pipe flow

From which $\frac{dv}{dy} \propto \frac{1}{y}$, *and from equation* $l \propto \sqrt{1 - \frac{y}{R}}$

Then it is follows that, in pipe flow:

$$l = \kappa y \left(1 - \frac{y}{R}\right)^{1/2} \tag{9.14}$$

Equation 9.12 now becomes

$$\tau_o / \rho \kappa^2 y^2 = (dv/dy)^2$$

or

$$dv/dy = \frac{\sqrt{\tau_o/\rho}}{\kappa y} = \frac{v_*}{\kappa y} \tag{9.15}$$

Attempts to represent the Reynolds stress have only been partly successful; this area represents one of the classic unsolved problems of fluid flow. As derived in Section 7.2, Prandtl mixing length theory is a plausible and practical, although not theoretically rigorous, approach. It is used here to derive the velocity distribution and other quantities because the results obtained are close to experimental data and the process gives insight to the physics of the flow.

Which illustrates again the pervasive presence of the friction velocity v_*. Integrating the last equation, one has:

$$v = \frac{v_*}{\kappa} \ln y + C \qquad (9.16)$$

There are two ways to approach evaluation of this unknown constant C.

*- First, if **$v = v_c$** at y = R (on the centerline), then and*

C = v_c — (v_*/K) lnR The result is: $\frac{v_c - v}{v_*} = -\frac{1}{K} ln \frac{y}{R}$

Second, one can try to find C in terms of the no-slip condition (v = 0) air = 0. But at y = 0, v = - ∞, according t o this Eqn. This is at least unrealistic (and not unexpected because the neglected viscous stresses dominate at the boundary)! It must be that the "turbulent" profile is replaced by a viscous-dominated profile near the wall and that there is some appropriate distance from the wall which marks the onset of this process- distance y' (nominal sublayer thickness δ_v, ($\delta_v = y'$).

Now very near the wall in a laminar flow, the velocity profile is linear, and if we match a linear laminar flow velocity profile with the turbulent profile at some distance y' from the boundary, and if we include additional experimental data from Nikuradse work, we come to the general equation of the velocity profile for turbulent flow in smooth pipes in the form:

$$\frac{v}{v_*} = 2.5 \ln \frac{v_* y}{\nu} + 5.5 \qquad (9.17)$$

Or $\frac{v}{v_*} = 5.75 log \frac{v_* y}{\nu} + 5.5$

Equating the equation $\frac{v_c - v}{v_*} = -\frac{1}{K} ln \frac{y}{R}$*, and the equation for laminar shear stress near the wall, we have:* $\frac{v}{v_*} = \frac{v_* y}{\nu}$

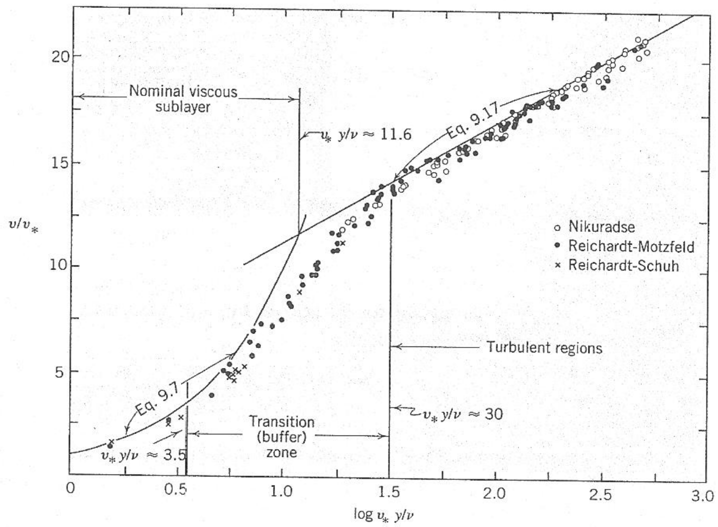
in smooth pipes (5 X 105 < R < 3 X 106) and in pipes with a uniform sand grain roughness showed that all velocity profiles (not just in smooth pipes) could be characterized by the single equation

Therefore, it must be that dv/dy l/y and, from Eqn. 9.12, (y/R). Near the wall = Ky it follows that, in a pipe flow, (9.14) Equation 9.12 now becomes dy/dy

We can find the intersection of both where velocity is the same at which y=y'=δ_v, nominal sublayer thickness which gives:

$$\frac{v_* \delta_v}{\upsilon} = 11.6 \qquad or\ \delta_v = 11.6\frac{\upsilon}{v_*}$$

These results are illustrated, together with some experimental data in the following figure.



The flow rate in turbulent pipe flow is given:

$$Q = \int_0^R v(2\pi r\, dr) = 2\pi v_* \int_0^R \left[5.75 \log\left(\frac{R-r}{e}\right) + 8.5\right] r\, dr$$

yielding

$$V = Q/\pi R^2 = v_* \left[5.75 \log\frac{R}{e} + 4.75\right]$$

$$\frac{V}{v_*} = 5.75 log\frac{v_* R}{\upsilon} + 1.75 \qquad for\ smooth\ pipe\ and$$

From which,

$$\frac{v_c}{V} = 1 + 4.07\sqrt{\frac{f}{8}} \qquad also\ \ for\ smooth\ pipes$$

Which is adjusted to conform with experiment.

Fig. 4.5 gives a comparison (for the same flow rate and mean velocity) of typical velocity profiles for laminar and turbulent pipe flow.

Next it is possible to developed a relation between f and Re, as results

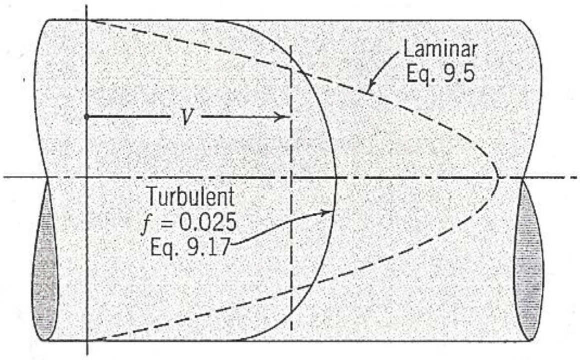


Fig. 9.5

$$\boxed{\frac{1}{\sqrt{f}} = 2.0 \log(\mathbf{R}\sqrt{f}) - 0.8} \qquad \text{(Smooth pipes)} \qquad (9.21)$$

Where **$Re = V\,d/\,v$** *is the pipe Reynolds number.*

Also we can reach the following relation for laminar sublayer:

$$\frac{\delta_v}{d} = \frac{11.6\upsilon}{v_* d} = \frac{11.6\upsilon}{vd\sqrt{\dfrac{f}{8}}} = \frac{32.8}{Re\sqrt{f}}\text{, which gives } Re\sqrt{f} = \frac{32.8}{(\delta_v/d)}$$

Then substituting in the right hand side of relation between f and Re we have: $\dfrac{1}{\sqrt{f}} = 2.0\log\left[32.8/(\delta_v/d)\right] - 0.8$

Which gives, that for turbulent flow over smooth wall the friction factor is a function only of the ratio of the sublayer thickness to the pipe diameter.

PROBLEM 9.4

Water at 20°C flows in a 75-mm diameter smooth pipeline. According to a wall shear meter (Section 14.11), $\tau_o = 3.68 \text{ N/m}^2$. Calculate the thickness of the viscous sublayer, the friction factor, the mean velocity and flowrate, the centerline velocity, the shear stress and velocity 25 mm from the pipe centerline, and the head lost in 1 000 m of this pipeline.

Solution:

Because the friction velocity appears in so many of the relevant equations, we will calculate that quantity first. By definition,

$$v_* = \sqrt{\frac{\tau_o}{\rho}} = \sqrt{\frac{3.68 \text{ N/m}^2}{998 \text{ kg/m}^3}} = 0.061 \text{ m/s}$$

Now we employ Eq. 9.18 to compute sublayer thickness.

$$\delta_v = 11.6(\nu/v_*) = 11.6(1 \times 10^{-6} \text{ m}^2/\text{s})/(0.061 \text{ m/s}) \qquad (9.18)$$

$$= 1.9 \times 10^{-4} \text{ m} = 0.19 \text{ mm} \bullet$$

PROBLEM 9.4

Water at 20°C flows in a 75-mm diameter smooth pipeline. According to a wall shear meter (Section 14.11), $\tau_o = 3.68 \text{ N/m}^2$. Calculate the thickness of the viscous sublayer, the friction factor, the mean velocity and flowrate, the centerline velocity, the shear stress and velocity 25 mm from the pipe centerline, and the head lost in 1 000 m of this pipeline.

solution

Because the friction velocity appears in so many of the relevant equations, we will calculate that quantity first. By definition,

$$v_* = \sqrt{\frac{\tau_o}{\rho}} = \sqrt{\frac{3.68 \text{ N/m}^2}{998 \text{ kg/m}^3}} = 0.061 \text{ m/s}$$

Now we employ Eq. 9.18 to compute sublayer thickness.

$$\delta_v = 11.6(\nu/v_*) = 11.6(1 \times 10^{-6} \text{ m}^2/\text{s})/(0.061 \text{ m/s}) \qquad (9.18)$$

$$= 1.9 \times 10^{-4} \text{ m} = 0.19 \text{ mm} \bullet$$

We turn to Eq. 9.23 to calculate the friction factor.

$$\frac{1}{\sqrt{f}} = 2.0 \log[32.8/(\delta_v/d)] - 0.8 \qquad (9.23)$$

$$= 2.0 \log[32.8/(0.19 \text{ mm}/75 \text{ mm})] - 0.8 = 7.42$$

$$f = 0.018 \bullet$$

Equation 9.19 will be used to compute the average velocity although Eq. 9.4 would do as well.

$$\frac{V}{v_*} = 5.75 \log \frac{v_* R}{\nu} + 1.75 \qquad (9.19)$$

$$= 5.75 \log \frac{0.061 \text{ m/s} \times 0.037\,5 \text{ m}}{1 \times 10^{-6} \text{ m}^2/\text{s}} + 1.75 = 21.1$$

$$V = 1.29 \text{ m/s} \bullet$$

$$Q = VA = 1.29 \text{ m/s} \times (\pi/4) \times (0.075 \text{ m})^2 \doteq 0.005\,7 \text{ m}^3/\text{s} \bullet$$

To compute the centerline velocity, we utilize Eq. 9.20.

$$\frac{v_c}{V} = 1 + 4.07\sqrt{f/8} = 1 + 4.07\sqrt{.018/8} = 1.193 \tag{9.20}$$

$$v_c = 1.54 \text{ m/s} \bullet$$

We know that shear stress varies linearly with radius. The point in question is 25 mm from the pipe centerline or 2/3 of the way to the wall. As a consequence, the shear stress at this point is 2/3 of the value at the wall or 2.45 N/m^2. •

Equation 9.17 can be used to find the velocity at this point (remember, y is measured outward from the wall).

$$\frac{v}{v_*} = 5.75 \log \frac{v_* y}{\nu} + 5.5 = 5.75 \log \frac{0.061 \text{ m/s} \times .012\,5 \text{ m}}{1 \times 10^{-6} \text{ m}^2/\text{s}} + 5.5 = 22.1 \tag{9.17}$$

$$v = 1.35 \text{ m/s} \bullet$$

Finally, we calculate the head loss in 1 000 m of pipe using the Darcy-Weisbach formula.

$$h_L = f\frac{l}{d}\frac{V^2}{2g_n} = 0.018 \frac{1\,000 \text{ m}}{0.075 \text{ m}} \frac{(1.29 \text{ m/s})^2}{2 \times 9.81} = 20.4 \text{ m} \bullet \tag{9.2}$$

Before the development of the foregoing generalizations by Prandtl, von Kármán, and Nikuradse, a pioneering effort was made by Blasius to relate velocity profile, wall shear, and friction factor for turbulent flow in smooth pipes. Although Blasius's work has been superseded by these generalizations, it is still of some importance in engineering (in spite of its limited scope and empiricism) because of a mathematical simplicity which allows easy visualization and leads directly to useful (but approximate) results.

First, Blasius[3] showed that the curve representing the friction factor (for 3 000 < $\mathbf{R}$ < 100 000) could be closely approximated by the equation

$$\boxed{f = \frac{0.316}{\mathbf{R}^{0.25}}} \quad \text{(Blasius)} \tag{9.24}$$

When this is substituted in the Darcy-Weisbach Eq. 9.2, it is noted that $h_L \propto V^{1.75}$ for the turbulent flow in smooth pipes with $\mathbf{R} < 10^5$.

Substituting Eq. 9.24 into Eq. 9.3 produces

$$\tau_o = \frac{0.316}{(2RV\rho/\mu)^{0.25}} \frac{\rho V^2}{8} = 0.033\,2\mu^{1/4}R^{-1/4}V^{7/4}\rho^{3/4} \tag{9.25}$$

Blasius then assumed that the turbulent velocity profile could be approximated (see Fig. 9.6) by a power relationship

$$\frac{v}{v_c} = \left(\frac{y}{R}\right)^m$$

For this equation the mean velocity V may be related to the center velocity v_c by applying Eq. 4.8:

$$V\pi R^2 = \int_R^0 v_c \left(\frac{y}{R}\right)^m 2\pi(R - y)(-dy)$$

which gives

$$\frac{V}{v_c} = \frac{2}{(m+1)(m+2)} \tag{9.26}$$

from which, by substitution into the profile equation, we derive

$$V = \frac{2}{(m+1)(m+2)} v \left(\frac{R}{y}\right)^m$$

Substituting this expression for V into Eq. 9.25,

$$\tau_o = 0.033\,2 \left[\frac{2}{(m+1)(m+2)}\right]^{7/4} \mu^{1/4} R^{-1/4+(7m/4)} v^{7/4} y^{-(7m/4)} \rho^{3/4}$$

However (Blasius reasoned), wall shear τ_o could depend only on the form of the velocity profile and the physical properties of the fluid but could not be affected by pipe size R; thus the exponent of R must be zero, and from this the exponent m must be equal to $1/7$. Validation of this depended on experimental measurements of turbulent velocity profiles

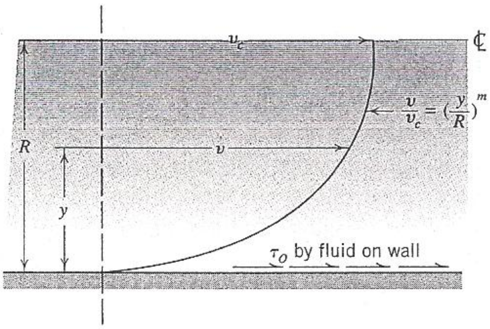


Fig. 9.6

which were found to agree quite well with the hypothesis above; thus the so-called *seventh-root law* for turbulent velocity distribution has been widely accepted. It is written

$$\boxed{\frac{v}{v_c} = \left(\frac{y}{R}\right)^{1/7}} \tag{9.27}$$

A useful corollary of this law is an equation for wall shear τ_o in terms of v_c and R. Using $m = 1/7$ in Eq. 9.26 yields $V/v_c = 49/60$, and, substituting $49v_c/60$ into Eq. 9.25 for V and rearranging,

$$\tau_o = 0.046\,4 \left(\frac{\mu}{v_c \rho R}\right)^{1/4} \frac{\rho v_c^2}{2} \tag{9.28}$$

which will be used later in the approximate analysis of turbulent boundary layers.

PROBLEM 9.5

For the conditions of Illustrative Problem 9.4, calculate the friction factor, wall shear stress, centerline velocity, and the velocity 25 mm from the pipe centerline using the seventh-root law.

Solution:

The friction factor can be calculated from Eq. 9.24. However, we will first calculate the Reynolds number **R**.

$$\mathbf{R} = \frac{Vd}{\nu} = \frac{1.29 \text{ m/s} \times 0.075 \text{ m}}{1 \times 10^{-6} \text{ m}^2/\text{s}} = 96\,750$$

Checking to see if we are in the range of applicability of the Blasius approximation, we note that 96 750 is greater than 3 000 and less than 100 000; so, the Blasius approach should yield good results.

Now, calculating the friction factor,

$$f = \frac{0.316}{\mathbf{R}^{0.25}} = \frac{0.316}{(96\,750)^{0.25}} = 0.018 \bullet \tag{9.24}$$

Before calculating wall shear stress, we must find the centerline velocity. We go to Eq. 9.26.

$$\frac{V}{v_c} = \frac{2}{\left(\frac{1}{7} + 1\right)\left(\frac{1}{7} + 2\right)} = \frac{49}{60} \quad \text{so} \tag{9.26}$$

$$v_c = \frac{60}{49} V = \frac{60}{49} \times 1.29 \text{ m/s} = 1.58 \text{ m/s} \bullet$$

which were found to agree quite well with the hypothesis above; thus the so-called *seventh-root law* for turbulent velocity distribution has been widely accepted. It is written

$$\boxed{\frac{v}{v_c} = \left(\frac{y}{R}\right)^{1/7}} \tag{9.27}$$

A useful corollary of this law is an equation for wall shear τ_o in terms of v_c and R. Using $m = 1/7$ in Eq. 9.26 yields $V/v_c = 49/60$, and, substituting $49v_c/60$ into Eq. 9.25 for V and rearranging,

$$\tau_o = 0.046\,4 \left(\frac{\mu}{v_c \rho R}\right)^{1/4} \frac{\rho v_c^2}{2} \tag{9.28}$$

which will be used later in the approximate analysis of turbulent boundary layers.

4.3.2- ROUGH PIPES

Pipe friction in rough pipes at high Reynolds numbers will be governed primarily by the size and pattern of the roughness, since disruption of the viscous sublayer will render any laminar flow action in that region ineffective. However, experiments show, as noted above, that the logarithmic velocity profile given by $\frac{v_c - v}{v_*} = -\frac{1}{K} \ln \frac{y}{R}$, *is applicable for both smooth and rough pipes.*

A development similar to that used for smooth pipe flow, except with the average roughness height e used instead of the viscosity, and tempered by the experimental work of Nikuradse on pipes with sand grain roughness, yields the following equation for the velocity profile.

$$\boxed{\frac{v}{v_*} = 5.75 \log \frac{y}{e} + 8.5} \quad \text{(Rough pipes)} \qquad (9.29)$$

As before the flowrate Q is found by integration:

$$Q = \int_0^R v(2\pi r dr) = 2\pi v_* \int_0^R \left[5.75 \log\left(\frac{R-r}{e}\right) + 8.5 \right] r dr, \quad \textit{yielding to}$$

$$V = \frac{Q}{\pi R^2} = v_* \left[5.75 \log \frac{R}{e} + 4.75 \right], \textit{thus} \quad \frac{V}{v_*} = 5.75 \log \frac{R}{e} + 4.75 \quad \textit{one}$$

can used this equation to verify that $\frac{v_c}{V} = 1 + 4.07\sqrt{\frac{f}{8}}$, *which is clearly, then valid for all turbulent flows cases.*

Using $v_* = V\sqrt{f/8}$, *in the previous equation this gives:*

$\frac{1}{\sqrt{f}} = 2.03 \log \frac{Re}{e} + 1.68$, *this equation is adjusted by help of Nikuradse experimental results, and the final equation is:*

$$\frac{1}{\sqrt{f}} = 2.0 \log \frac{Re}{e} + 1.75 \quad \textit{Or} \quad \frac{1}{\sqrt{f}} = 2.0 \log \frac{d}{e} + 1.14$$

For turbulent flow in rough pipes the friction factor is a function only of the relative roughness e/R or e/d and is not a function of Reynolds number (compare Eqn. 4.31 to Eqn. 4.21). Friction effects

are produced in fully rough flow by roughness alone, without dependence on viscous action.

PROBLEM "REPORT"

The mean velocity in a 300-mm pipeline is 3 m/s. The relative roughness of the pipe is 0.002 and the kinematic viscosity of the water is 9 X 10^{-7} m^2/s. Determine the friction factor, the centerline velocity, the velocity 50 mm from the pipe wall, and the head lost in 300 m of this pipe under the assumption that the pipe is rough.

4.4- FRICTION FACTOR

4.4.1- CIRCULER PIPE

The evaluation of pipe friction factors is extensively based on experimental work so it is informative to approach the representation of the friction factor from the standpoint of dimensional analysis and similitude. From the analyses presented in previous sections, it is known that the wall shear stress $\boldsymbol{\tau_o}$ *depends on the mean velocity V, the pipe diameter d9 the mean height of the wall roughness projections e, the fluid density ρ, and the fluid viscosity /x. Thus*

$$\phi(\tau_o, V, d, e, \rho, \mu) = 0$$

Using the Il-theorem (Section 8.2), with V, d, and p as the repeating variables, gives

$$\Pi_1 = \frac{\tau_o}{\rho V^2} \qquad \Pi_2 = \frac{Vd\rho}{\mu} \qquad \Pi_3 = \frac{e}{d}$$

Setting j as a function of U2 and n3, we obtain

$$\tau_o = \rho V^2 \phi'\left(\frac{Vd\rho}{\mu}, \frac{e}{d}\right) = \rho V^2 \phi'\left(\mathbf{R}, \frac{e}{d}\right) = \frac{\rho V^2}{8}\, \phi''\left(\mathbf{R}, \frac{e}{d}\right)$$

Comparison with equation 4.3 shows that

$$f = \phi''\left(\mathbf{R}, \frac{e}{d}\right) \tag{9.38}$$

Thus, the dimensional analysis approach confirms the experimental results and previous analyses that the friction factor depends only on the Reynolds number of the flow and the pipe relative roughness e/d. The physical significance of Eqn. 4.38 may be stated briefly as the friction factors of a number of pipes will be the same if their Reynolds

numbers, roughness patterns, and relative roughness are the same. When this is interpreted by the laws of similitude, its basic meaning is the friction factors of the pipes are the same if their flow pictures in every detail are geometrically and dynamically similar.

Eqn 4.38 suggests a convenient means of presenting the experimental data on the friction factor. This was first used by Blasius in 1913 and by Stanton6 in 1914 and consists of a logarithmic plot of friction factor versus Reynolds number, with relative roughness as the parameter. Figure 4.9 shows Nikuradse's data plotted in the Blasius-Stanton format, revealing the relationship between, f , R, and e/d. Recall that Nikuradse's experiments employed fixing sand grains on the inside of the pipe to establish an easily measurable index of roughness. The purpose was to demonstrate conclusively the inter-relationship among, R, and e/d. in that regard, the effort was successful and the following important fundamentals were clearly established.

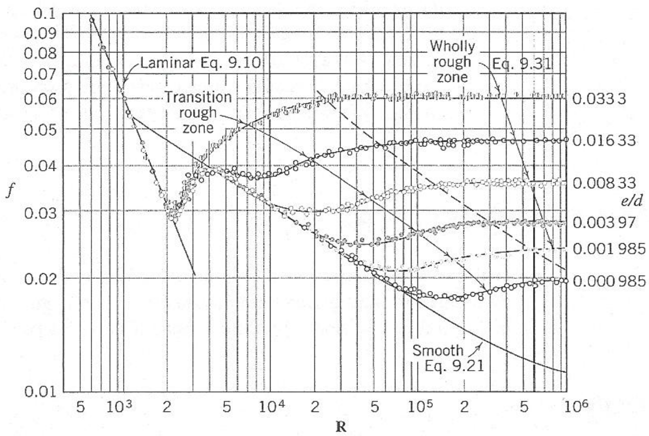


Fig. 9.9 Blasius-Stanton diagram with Nikuradse's experimental data.

1.The physical difference between the laminar and turbulent flow regimes is indicated by the change in the relationship of f to R near the critical Reynolds number of 2100.

2. the laminar regime is characterized by a single curve, given by the equation = 64/R for all surface roughness. This confirms that head loss in laminar flow is independent of surface roughness and that hL V.

3. in turbulent flow a curve of versus R exists for every relative roughness, e/d9 and the horizontal aspect of the curves confirms that for rough pipes the roughness is more important than the Reynolds number in determining the magnitude of the friction factor.

4.At high Reynolds numbers the friction factors of rough pipes become constant, dependent wholly on the roughness of the pipe, and thus independent of the Reynolds number.

5.Although the lowest curve was obtained from tests on hydraulically smooth pipes, many of Nikuradse's rough pipe test results coincide with it for 5000 < R < 50000. Here the roughness is submerged in the viscous sublayer (Sections 4.3 and 4.5) and can have no effect on friction factor and head loss, which depend on viscosity effects alone. From Darcy-Weisbach equation h V175 for turbulent flow in smooth pipes where the friction factor is given by the Blasius expression (Eqn. 4.24).

6.The series of curves for the rough pipes diverges from the smooth pipe curve as the Reynolds number increases. In other words, pipes that are smooth at low values of R become rough at high values of R. This is explained by the thickness of the viscous sublayer decreasing as the Reynolds number increases, thus exposing smaller roughness protuberances to the turbulent region and causing the pipe to exhibit the properties of a rough pipe.

MOODY DIAGRAM

Earlier, Colebrook showed that Nikuradse's results were not representative of commercial pipes. Roughness patterns and variations in the height of roughness projections in commercial pipes resulted in friction factors which were considerably different than Nikuradse's in the transition zone between smooth and wholly rough turbulent flow. Benedict gives a particularly lucid discussion of this apparent discrepancy. He notes that the work of Colebrook and others on commercial pipe rather conclusively demonstrates that the friction factor varies as described by Eqn. 4.34 rather than as shown in Fig. 4.9. Later, Moody presented the equations of Colebrook in graphical form using the Blasius-Stanton format. The result of this work is shown M Fig. 4.10. This plot, known as the Moody diagram, along with the e-values for commercial pipes given in Fig. 4.11, can be used directly in the solution of engineering problems. A computer program, DARCY, which calculates a friction factor from the Moody diagram, is given in Appendix 7. However, a word of caution is necessary. The roughness of commercial pipe materials varies widely with the manufacturer, with years in service, and with liquid conveyed. Corrosion of pipe wall

material and deposition of scale, slime, and such can drastically increase the roughness of the pipe and the resulting friction factor. One of the practical considerations of pipe flow calculations, which pose a difficult problem for the engineer, is the prediction of friction factors which will actually be realized after a pipeline is constructed. Extensive practical experience is a valuable asset in his situation.

Example 4.10

Water at 100°F flows in a 3 inch pipe at a Reynolds number of 80 000. This is a *commercial pipe* with an equivalent sand grain roughness of 0.006 in. What head loss is to be expected in 1 000 ft of this pipe?

Note that this is the same problem as Illustrative Problem 9.8 except we are using commercial pipe.

SOLUTION

We will use the Moody diagram to find the friction factor but first we must calculate the relative roughness e/d.

$$\frac{e}{d} = \frac{0.006 \text{ in.}}{3.00 \text{ in.}} = 0.002$$

With $\mathbf{R} = 80\,000$ and $e/d = 0.002$, we go to the Moody diagram of Fig. 9.10 and find

$$f \cong 0.025\,5$$

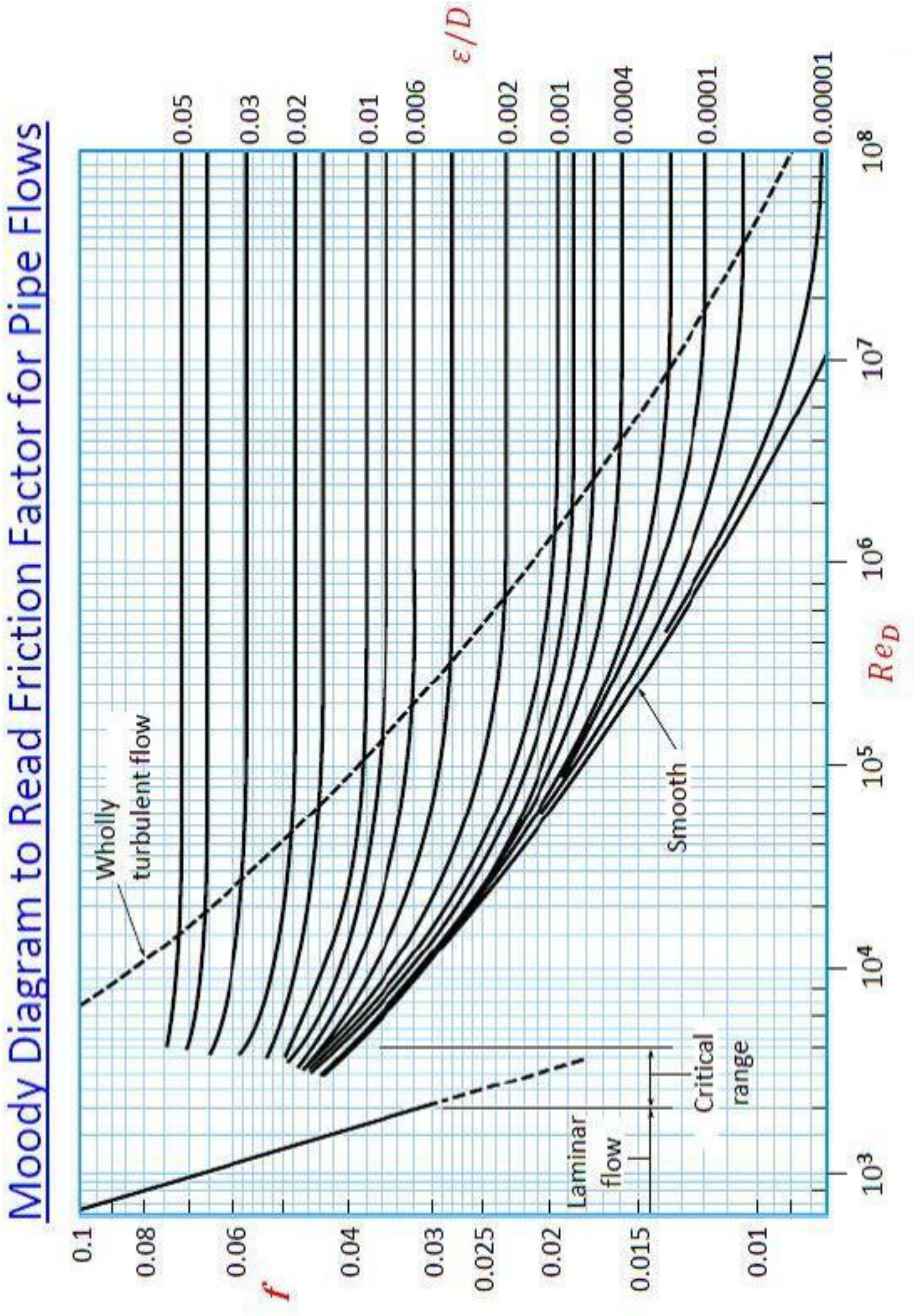
From Illustrative Problem 9.8, $V = 2.36$ ft/s. We can now compute the head loss using the Darcy-Weisbach equation.

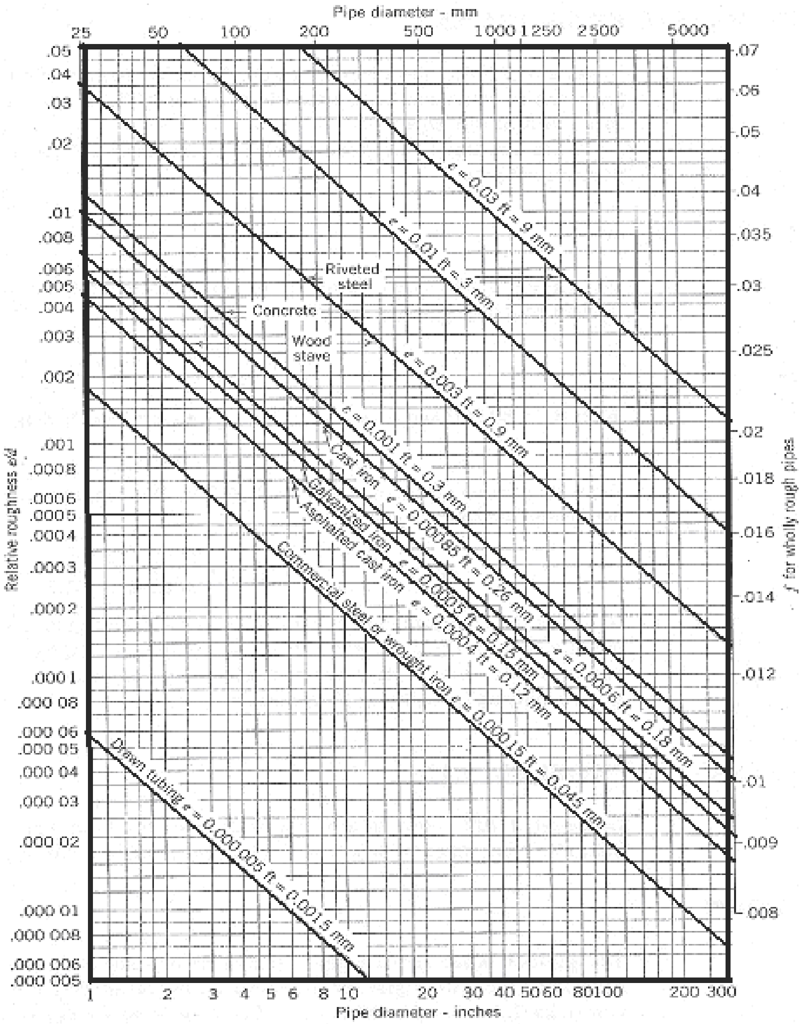
$$h_L = f\frac{l}{d}\frac{V^2}{2g_n} = 0.025\,5\,\frac{1\,000}{3/12}\,\frac{2.36^2}{2 \times 32.2} = 8.8 \text{ ft} \qquad (9.2)$$

Note that the commercial pipe with an *equivalent* sand grain roughness equal to that of a pipe lined with real sand grains 0.006 inches in diameter has a 30% higher head loss at this Reynolds number. However, both of these pipes would have the same head loss under smooth pipe and wholly rough conditions.

EXAMPLE

Crude oil at 20°C flows in a riveted steel pipe 1.00 m in diameter at a mean velocity of 2.0 m/s. What range of head loss is to be expected in 1 km of pipeline?


Moody Diagram to Read Friction Factor for Pipe Flows
ε/D
0.05
0.03
0.02
0.01
0.006
0.002
0.001
0.0004
0.0001
0.00001
Wholly turbulent flow
Smooth
Laminar flow
Critical range
f
0.1
0.08
0.06
0.04
0.03
0.025
0.02
0.015
0.01
Re_D
10^3
10^4
10^5
10^6
10^7
10^8



SOLUTION

The question posed above suggests there may be some uncertainty in the evaluation of the head loss. We will begin our investigation by calculating the Reynolds number. From Appendix 2, crude oil at 20°C has a density $\rho = 855.6\ \text{kg/m}^3$ and an absolute viscosity $\mu = 71.8 \times 10^{-4}\ \text{Pa·s}$.

$$\mathbf{R} = \frac{Vd\rho}{\mu} = \frac{2.0\ \text{m/s} \times 1.00\ \text{m} \times 855.6\ \text{kg/m}^3}{71.8 \times 10^{-4}\ \text{Pa·s}} = 2.4 \times 10^5$$

Now, to find the relative roughness, we go to Fig. 9.11 for riveted steel pipe 1 m in diameter and read

$$\frac{e}{d} = 0.000\,9 \text{ to } 0.009$$

So now we know what the uncertainty is about. Apparently there are several ways to construct riveted steel pipe with the number, type and spacing of rivets open to question. We will take the two extreme values and find the head loss for each. From the Moody diagram of Fig. 9.10,

$$\frac{e}{d} = 0.000\,9 \qquad f = 0.020\,5 \quad \text{(By program DARCY, App. 7, } f = 0.020\,4\text{)}$$

$$\frac{e}{d} = 0.009 \qquad f = 0.036\,5 \quad \text{(By program DARCY, App. 7, } f = 0.036\,5\text{)}$$

The Darcy-Weisbach equation 9.2 yields

$$h_L = f\frac{l}{d}\frac{V^2}{2g_n} = 0.020\,5\,\frac{1\,000}{1}\,\frac{2^2}{2\times 9.81} = 4.2\text{ m} \bullet$$

$$h_L = f\frac{l}{d}\frac{V^2}{2g_n} = 0.036\,5\,\frac{1\,000}{1}\,\frac{2^2}{2\times 9.81} = 7.4\text{ m} \bullet$$

The range of head loss is 4.2 m to 7.4 m per km.

4.4.1-PIPE FRICTION IN NONCIRCULAR PIPES THE HYDRAULIC RADIUS:

Although the majority of pipes used in engineering practice are of circular cross section, occasions arise when calculations must be carried out on head loss in rectangular ducts and other conduits of noncircular form. The foregoing equations for circular pipes may be adapted to these special problems through use of the hydraulic radius concept.

The hydraulic radius, RHI is defined as the area, A, of the flow cross section divided by its "wetted perimeter," P. In a circular pipe of diameter d,

$$R_h = \frac{d}{4} \qquad \text{or} \qquad d = 4R_h \tag{9.39}$$

This value may be substituted into the Darcy-Weisbach equation for head loss and into the expression for the Reynolds number with the following results:

$$h_L = \frac{f}{4}\frac{l}{R_h}\frac{V^2}{2g_n} \tag{9.40}$$

$$\mathbf{R} = \frac{V(4R_h)\rho}{\mu} \tag{9.41}$$

from which the head loss in many conduits of noncircular cross section may be calculated with the aid of the Moody diagram of Fig. 4.10.

The calculation of lost head in noncircular conduits thus involves the calculation of the hydraulic radius of the flow cross section and the use of the friction factor obtained for an equivalent circular pipe having a diameter d equal to $4R_H$. In view of the complexities of viscous sublayer, turbulence, roughness, shear stress, and so forth, it seems surprising at first that a circular pipe equivalent to a noncircular conduit may be obtained so easily, and it would, therefore, be expected that the method might be subject to certain limitations.

The method gives satisfactory results when the problem is one of turbulent flow but, if it is used for laminar flow, large errors are introduced.

The foregoing facts may be justified analytically by examining further the structure of Eqn. 4.40 in which hL varies with l/R_h. From the definition of the hydraulic radius, its reciprocal is the wetted perimeter per unit of flow cross section and is, therefore, an index of the extent of the boundary surface in contact with the flowing fluid. The hydraulic radius may be safely used in the equation above when resistance to flow and head loss are primarily dependent on the extent of the boundary surface, as for turbulent flow in which pipe friction phenomena are confined to a thin region adjacent to the boundary surface and thus vary with the size of this surface. However, severe deviation from a circular flow cross section will prevent the hydraulic radius from accounting for changes in head loss, even for cases of turbulent flow. Tests on turbulent flow through annular passages,10 for example, show a large increase in friction factor with increase of the ratio of core diameter to pipe diameter.

In laminar flow, friction phenomena result from the action of viscosity throughout the whole body of the flow, are independent of roughness, and are not primarily associated with the region close to the boundary walls. In view of these facts the hydraulic radius technique cannot be expected to give reliable conversions from circular to noncircular passages in laminar flow. This expectation is borne out both by experiment and by analytical solutions of laminar flow in noncircular passages.

EXAMPLE.11

Calculate the loss of head and the pressure drop when air at an absolute pressure of 101.3 kPa and 15°C flows through 600 m of 450 mm by 300 mm smooth rectangular duct with a mean velocity of 3 m/s.

SOLUTION

We will treat this problem in the conventional fashion substituting $4R_h$ for d in the formulas. Since the duct is smooth, we will only need the Reynolds number to find the friction factor. From Appendix 2, we find values of $\rho = 1.225\ \text{kg/m}^3$ and $\mu = 1.789 \times 10^{-5}$ Pa·s for air at the specified pressure and temperature. The hydraulic radius for the duct is

$$R_h = \frac{\text{Area}}{\text{Wetted perimeter}} = \frac{0.45\ \text{m} \times 0.30\ \text{m}}{2 \times 0.45\ \text{m} + 2 \times 0.30\ \text{m}} = 0.090\ \text{m}$$

Now, we can compute the Reynolds number from Eq. 9.41.

$$\mathbf{R} = \frac{V(4R_h)\rho}{\mu} = \frac{3\ \text{m/s} \times (4 \times 0.090\ \text{m}) \times 1.225\ \text{kg/m}^3}{1.789 \times 10^{-5}} = 73\,950 \qquad (9.41)$$

From the smooth pipe line on the Moody diagram of Fig. 9.10,

$$f \cong 0.019$$

Equation 9.40 can be used to calculate the head loss in 600 m of the pipe.

$$h_L = f\frac{l}{4R_h}\frac{V^2}{2g_n} = 0.019\frac{600}{4 \times 0.090}\frac{3^2}{2 \times 9.81} = 14.5\ \text{m of air} \qquad (9.40)$$

The pressure drop is found from the equation

$$\Delta p = \gamma h_L = \rho g h_L = 1.225 \times 9.81 \times 14.5 = 174\ \text{Pa}$$

4.4.3- PIPE FRICTION-EMPIRICAL FORMULAS

The Darcy-Weisbach equation 4.2 has provided a rational basis for the analysis and computation of head loss. However, historically, a number of empirical formulas have been and still are being used for pipe friction calculations in engineering practice. A treatment of pipe friction would be incomplete without mentioning at least two of these formulas and how they relate to the Darcy-Weisbach equation.

By far the most widely used of these empirical formulations is the Hazen-Williams11 formula. It was developed to permit calculating the capacity of pipes to convey water.

(U.S. Customary units)	$V = 1.318 C_{hw} R_h^{0.63} S^{0.54}$	(9.42a)
(SI units)	$V = 0.849 C_{hw} R_h^{0.63} S^{0.54}$	(9.42b)

Where, Rh is the hydraulic radius, S is the head loss per unit length h_L/U and is a roughness coefficient associated with the pipe material. Table 1 gives some typical values of the Hazen-Williams coefficient.

TABLE 1 Hazen-Williams Coefficient C_{hw} and Manning n-values[a]

	C_{hw}	n
Extremely smooth pipes—PVC	150–160	0.009
Copper, aluminum tubing	150	0.010
Asbestos cement	140	0.011
New cast iron	130	0.013
Welded steel	130–140	0.012
Concrete	120–140	0.011–0.014
Ductile iron (cement lined)	140	0.011
Vitrified clay pipe	—	0.011–0.013
Riveted steel	110	0.013–0.017
Old cast iron	100	0.015–0.035

[a]These are typical values but, because of variabilities in fabrication, the user should consult the pipe manufacturer for recommended values of roughness coefficients.

Unlike the Darcy-Weisbach equation which can be applied to both laminar and turbulent flow over a wide range of fluids and temperatures, the Hazen-Williams formula is restricted to turbulent flow of water at normal temperature in a limited size range of relatively smooth pipes. However, the formula is so constructed that it permits direct calculation of the velocity of flow (or discharge) for a known allowable head loss. This feature is attractive in the design of water pipelines. Conversely, use of the Darcy-Weisbach equation in the same circumstances often requires a trial solution as one estimate the Reynolds number to find the friction factor and makes successive adjustments as one closes in on the answer.

In view of the formula configuration and the rather indefinite descriptions of the roughness coefficients, it is difficult to judge the range of validity of the Hazen-Williams formula without wide experience in its application. It is not clear whether are measure of absolute or relative roughness, whether there is any effect of Reynolds number in the formula, whether it applies only in smooth or rough pipe situations, etc. These questions can be answered fairly conclusively if the formula is rewritten in the form of the Darcy-Weisbach equation. Working with the U.S. Customary version, replacing Rh with d/4, S with h U and solving for h_L gives

$$h_L = \left[\frac{194}{C_{hw}^{1.85} (Vd)^{0.15} \; d^{0.015}} \right] \frac{l}{d} \frac{V^2}{2g_n}$$

If (V is multiplied by y>) 0, 15, a Reynolds number appears and the equation may be written as

$$h_L = \left[\frac{194}{\nu^{0.15} C_{hw}^{1.85}\ d^{0.015} \mathbf{R}^{0.15}}\right] \frac{l}{d} \frac{V^2}{2g_n}$$

Comparing this equation with the Darcy-Weisbach equation 4.2 and taking a nominal v-value of 0.00001 ft^2/s for water, the 4 'equivalent /-value for the Hazen-Williams formula is

$$\text{``}f\text{''} = \frac{1\,090}{C_{hw}^{1.85}\ d^{0.015} \mathbf{R}^{0.15}} \qquad (9.43)$$

The most effective means of visualizing the meaning of this value" is to plot the equation on the Moody diagram for several values. This is done on Fig. 4.12. The following conclusions can now be drawn in regard to the range of applicability of the Hazen-Williams formula

The Hazen-Williams formula is a "transition" formula working best as a smooth pipe formula or in the "early" transition zone between smooth and rough pipe flow. Note the = 150 smooth-pipe value applies fairly well over a range of Reynolds numbers from 100000 on but the plot suggests a value higher than 150, say 160, should be used for Reynolds numbers above 1000000. Also note that the formula grossly underestimates friction factors in smooth pipes for Reynolds numbers below 10000 (typical of trickle irrigation systems) and does not apply at all for laminar flow. Because the lines for the various values on Fig. 4.12 all have a constant slope, the Hazen-Williams formula should never be used for wholly rough pipes. More specifically, the Hazen-Williams formula should be used only

in zones where the pipe is relatively smooth and in the ' 'early'' part of its transition to rough flow. If there is a question as to applicability, it is always possible to estimate the Reynolds number of the flow and, with the proposed value, check Fig. 4.12 to verify the applicability of the formula.

2.Addressing the issue of relative roughness, we see it is clear from the above equation for that there is a small relative roughness effect built into the equation although the 0.015 exponent dampens the impact considerably. However, in the near-smooth'- zone where the Hazen-Williams formula applies, the effect of relative roughness is small, thereby minimizing the error.

3.There is definitely a strong Reynolds number effect built into the Hazen-Williams formula which renders it accurate in the "near-

smooth" transition range for Reynolds numbers greater than 100000 or with the appropriate value The second formula selected for examination is the Manning equation, noted for its application to open channel flow but also used for pipe flow. The Manning formulas for pipe flow are given as

$$\text{(U.S. Customary units)} \quad V = \frac{1.49}{n} R_h^{2/3} S^{1/2} \tag{9.44a}$$

$$\text{(SI units)} \quad V = \frac{1}{n} R_h^{2/3} S^{1/2} \tag{9.44b}$$

where Rh and S are as described for the Hazen-Williams formula and n is a roughness coefficient. Some typical values are given in Table 1.

We will investigate the range of applicability of the Manning formula in the same manner as was done with the Hazen-Williams formula. Again using the U.S. Customary version of the formula, and arranging in the Darcy-Weisbach configuration, we obtain.

$$``f" = \frac{185 n^2}{d^{1/3}} \tag{9.45}$$

It seems clear from this equation that there is no Reynolds number effect but there is a fairly strong relative roughness effect. This equation is plotted in Fig. 4.12 for two different n-values and two different diameters to show the effect of relative roughness. The following conclusions can be drawn regarding the applicability of the Manning formula.

1.There is no Reynolds number effect so the formula must be used only in the wholly rough flow zone where its horizontal slope can accurately match Darcy-Weisbach values provided the proper n-value is selected.

2.The relative roughness effect is correct in the sense that, for a given roughness, a larger pipe will have a smaller friction factor.

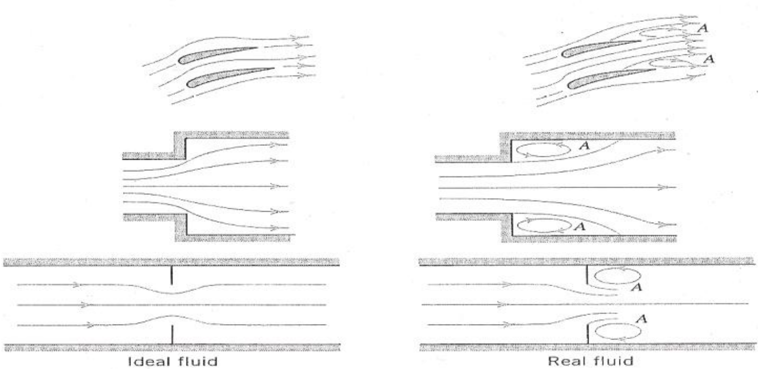
3.In a general sense, because the formula is valid only for rough pipes, the rougher the pipe, the more likely the Manning formula will apply.

Any other formula can be compared with the Darcy-Weisbach equation in the manner used above. For empirically-based formulas, this approach is strongly recommended unless the user is confident the formula is applicable to the case at hand.

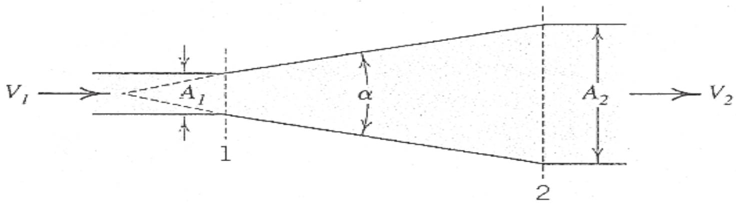
4.5- LOCAL LOSSES IN PIPELINES

Separation:

Separation has great impact on design and performance of certain familiar internal flow systems. For example, separation of the flow from the blades of a pump impeller 'or turbine rotor (runner) can cause. Careful design can prevent separation in these cases, but for flow through a valve or metering orifice or'-sudden expansion, separation is inevitable. Fig shows ideal and real fluids for the three cases.



The problem of separation is a major problem in a diffuser design (a classic internal flow). There is an optimum diffuser angle a - 6-8°, Fig.32.



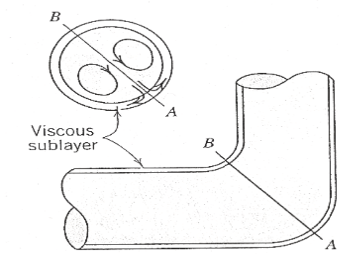
For greater angle, adverse pressure gradient increases which cause flow separation accompanied by great losses, while for small a great friction surface exist and shear stress is high accompanied also with great losses.

The "Axiom" mentioned before is also valid for internal flows, that is "Acceleration of real fluid tends to be efficient process, deceleration an' inefficient one". Accelerating motion through a convergent nozzle is accompanied by a favorable pressure gradient which stabilized the boundary layer and thus minimizes energy dissipation. Decelerating motion is accompanied by an adverse pressure gradient which "promotes separation, instability, eddy

formation and large energy dissipation. This axiom may be extended to explain the difference between the maximum efficiencies obtained in comparable complex machines such as hydraulic turbines and centrifugal pumps. In turbines the flow passage are predominately convergent and the flow is accelerated; in the pumps the situation is opposite. Hydraulic turbine efficiencies have been obtained up to 94%, but maximum centrifugal pump efficiencies are around 87%. The striking difference between these figures can be attributed primarily to the inherent efficiency and inefficiency of the acceleration and deceleration flow process, respectively.

4.6- SECONDARY FLOW

Let us consider the flow through a pipe bend, Fig. 4.33.



The fluid particles velocities at the center are the same, due to which arise centrifugal force which is in equilibrium with pressure force. The reduction of velocity at the outer part A, of the bend reduces the centrifugal force acting on particles moving near the wall.

As result of this, non-equilibrium between pressure and centrifugal forces, the twin eddy motion shown are developed, that is the secondary flow, when added to the main flow will cause a double spiral motion. The combination of double spiral flow and separation leads to increased head losses in bends, a large part of which may be caused by increased wall shear stress in and downstream from the bend because of the redistribution of the flow streamlines by the secondary flow.

Into the category of local losses in pipelines fall those losses incurred by change of cross section, bends, elbows, valves, and fittings of all types. Although in long pipelines these are distinctly minor losses and can often be neglected without serious error, in shorter pipelines an accurate knowledge of their effects must be known for correct engineering calculations.

The general aspects of local losses in pipelines may be obtained from a study of the flow phenomena about an abrupt obstruction placed in a pipeline (Fig. 4.13), which creates flow conditions typical of those which dissipate energy and cause local losses. Local losses usually result from rather abrupt changes (in magnitude or direction) of velocity; in general, increase of velocity (acceleration) is associated with small head loss but decrease of velocity (deceleration) causes large head loss because of the production of large-scale turbulence. In Fig. 4.13, useful energy is extracted to create eddies as the fluid decelerates between sections 2 and 3, and this energy is dissipated in heat as eddies decay between sections 3 and 4. Local losses in pipe flow are, therefore, accomplished in the pipe downstream from the source of the eddies, and the pipe friction processes in this length of pipe are complicated by the superposition of large-scale turbulence on the normal turbulence pattern.

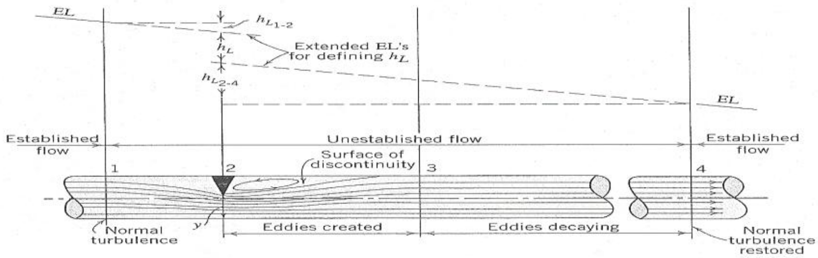


Fig. 9.13

Separate action of the normal turbulence and large-scale turbulence, although in reality a complex combination of the two processes exists. Assuming the processes independent allows calculation of the losses due to established pipe friction, h_{L2} and h_{Ll} and also permits the loss hL, due to the obstruction alone, to be assumed concentrated at section 2. This is a great convenience for engineering calculations since the total lost head in a pipeline may be obtained by simple addition of established pipe friction and local losses without detailed consideration of the above-mentioned complications.

Early experiments with water (at high Reynolds number) indicated that local losses vary approximately with the square of velocity and led to the proposal of the basic equation

$$\boxed{h_L = K_L \frac{V^2}{2g_n}} \qquad (9.46)$$

in which K_L, the loss coefficient, is, for a given flow geometry, practically constant at high Reynolds number; the loss coefficient tends

to increase with increasing roughness and decreasing Reynolds number,12 but these variations are usually of minor importance in turbulent flow. The magnitude of the loss coefficient is determined primarily by the flow geometry, that is, by the shape of the obstruction or pipe fitting.

When an abrupt enlargement of section (Fig. 4.14) occurs in a pipeline, a rapid de-celebration takes place, accompanied by characteristic large-scale turbulence, which may persist in the larger pipe for a distance of 50 diameters or more downstream before the normal turbulence pattern of established flow is restored. Simultaneous application of the continuity, Bernoulli, and momentum principles to this problem has shown that (with certain simplifying assumptions),

$$\boxed{h_L = K_L \frac{(V_1 - V_2)^2}{2g_n}} \qquad (9.47)$$

2Note that this is the same trend followed by the friction factor, in which K_L= 1.

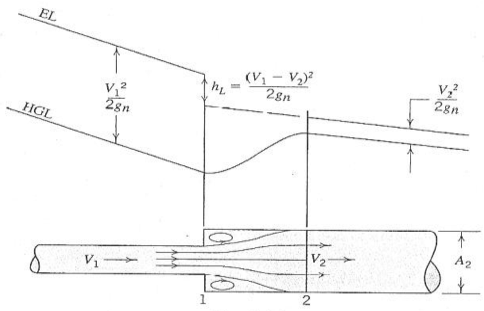


Fig. 9.14

Experimental determinations of K_L confirm this value within a few percent, making it quite adequate for engineering use. A special case of an abrupt enlargement exists when a (relatively small) pipe discharges into a (relatively large) tank or reservoir. Here the velocity downstream from the enlargement may be taken to be zero, and when the lost head (called the exit loss) is calculated from Eqn. 4.47 it is found to be the velocity head in the pipe.

The loss of head due to gradual enlargement is, of course, dependent on the shape of the enlargement. Tests have been carried out by Gibson13 on the losses in conical enlargements, and the results are expressed by Eqn.4.47, in which K_L is primarily dependent on the cone angle but is also a function of the area ratio, as shown in Fig. 4.15. Because of the large surface of the conical enlargement which contacts the fluid, the coefficient loss coefficients for conical enlargements.

embodies the effects of wall friction as well as those of large-scale turbulence. In an enlargement of small central angle, K_L will result almost wholly from surface friction; but, as the angle increases and the enlargement becomes more abrupt, not only is the surface area reduced but also separation occurs, producing large eddies, and here the energy dissipated in the eddies determines the magnitude of K_L. From the plot it may be observed that:

(1) There is an optimum cone angle of about 7° where the combination of the effects of surface friction and eddying turbulence is a minimum. (2) It is better to use a sudden enlargement than one of cone angle around 60°, since KL is smaller for the former.

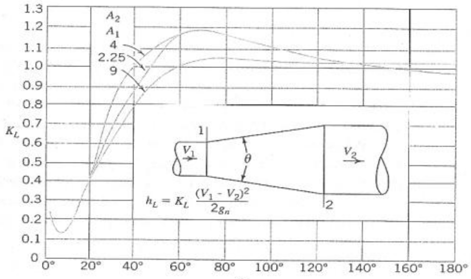


Fig. 9.15 Loss coefficients for conical enlargements.[12] (Source: A. H. Gibson, Hydraulics and its Applications, 4th ed., 1930.)

Gradual enlargements in passages (termed diffusers) of various forms are widely used in engineering practice for pressure recovery, that is, pressure rise in the direction of flow. No attempt is made here to review the extensive literature14 on diffusers, but it should be realized that Gibson's results cannot give a reliable solution to this problem. His tests were made, as are all such tests for local losses, with long straight lengths of pipe upstream and downstream from the enlargement (see Fig. 4.13). The designer, however, is frequently more interested in the pressure rise through the diffuser and substitution of a short nozzle for the upstream pipe length. The tests of Gibson and others have shown that the pressure will continue to rise for a few pipe diameters downstream from section 2, owing primarily to readjustment of velocity distribution from a rather pointed one caused by deceleration through the diffuser to the flatter one of turbulent flow.15 From this it may be concluded that pressure rise through the diffuser, computed from application of the Bernoulli equation with data from Fig. 4.15, will be larger than that which will actually be realized. Substitution of a nozzle for the upstream pipe length will alter the inlet velocity distribution from the standard one of turbulent flow to a practically uniform one with a thin boundary layer. The effect is to reduce the losses by

stabilizing the flow and delaying separation; not only do smaller loss coefficients result, but the cone angle for minimum losses is larger, thus allowing a shorter diffuser for the same area ratio.

EXAMPLE.13

A 300 mm horizontal water line enlarges to a 600 mm line through a 20° conical enlargement. When 0.30 m^3/s flow through this line, the pressure in the smaller pipe is 140 kPa. Calculate the pressure in the larger pipe, neglecting pipe friction.

SOLUTION

In determining the head loss in a conical enlargement, Eq. 9.47 applies with the K_L-value obtained from Fig. 9.15. First, we will calculate the velocities in each pipe.

$$V_{300} = \frac{Q}{A_{300}} = \frac{0.30 \text{ m}^3/\text{s}}{(\pi/4)(0.300 \text{ m})^2} = 4.24 \text{ m/s}$$

$$V_{600} = \frac{Q}{A_{600}} = \frac{0.30 \text{ m}^3/\text{s}}{(\pi/4)(0.600 \text{ m})^2} = 1.06 \text{ m/s}$$

From Fig. 9.15, $K_L = 0.43$. To compute the pressure in the large pipe, we turn to the work-energy equation 7.46 without pumps or turbines.

$$z_{300} + \frac{p_{300}}{\gamma} + \frac{V_{300}^2}{2g_n} = z_{600} + \frac{p_{600}}{\gamma} + \frac{V_{600}^2}{2g_n} + h_L \qquad (7.53)$$

where $h_L = K_L \dfrac{(V_{300} - V_{600})^2}{2g_n}$

Taking the datum as the pipe centerline eliminates z from the calculations leaving

$$\frac{140 \times 10^3 \text{ Pa}}{9\,800 \text{ N/m}^3} + \frac{(4.24 \text{ m/s})^2}{2 \times 9.81} = \frac{p_{600}}{\gamma} + \frac{(1.06 \text{ m/s})^2}{2 \times 9.81} + 0.43\,\frac{(4.24 - 1.06)^2}{2 \times 9.81}$$

$$\frac{p_{600}}{\gamma} = 14.6 \text{ m}$$

$$p_{600} = 14.6 \times 9\,800 = 143\,000 \text{ Pa} = 143 \text{ kPa} \bullet$$

This is the pressure to be expected a metre or so downstream from the end of the enlargement.

Flow through an abrupt contraction is shown in Fig. 4.16 and is featured by the formation of a vena contracta and subsequent deceleration and expansion of the live stream of flowing fluid.

Experimental measurements of K_L are somewhat conflicting in magnitude although they exhibit a well-established trend from 0.5 for A_2/A_l = 0 to 0 for A_2/Ax = 1. In view of this it is entirely adequate for engineering practice to use a synthesis of analytical approaches and generally accepted experimental information.16 The result is given in Table 2 where Cc = Ac/A2.

A square-edged pipe entrance (Fig. 4.17a) from a large body of fluid is the limiting case of the abrupt contraction, with A2/Al = 0 (A is virtually infinite here). The head loss is expressed by

Eqn. 4.46 in which KL is close to 0.5 for highly turbulent flow, as mentioned above.

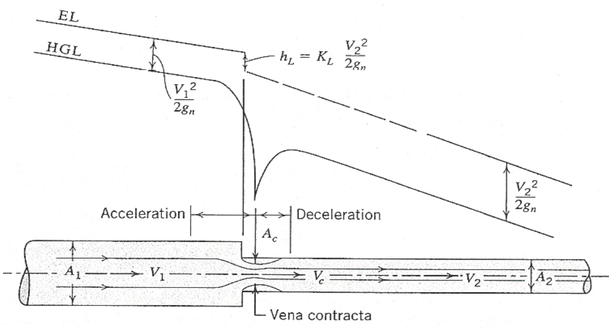


Fig. 9.16 Abrupt contraction.

TABLE 2 Contraction, & and Loss, A'A, Coefficients for Abrupt Contractions

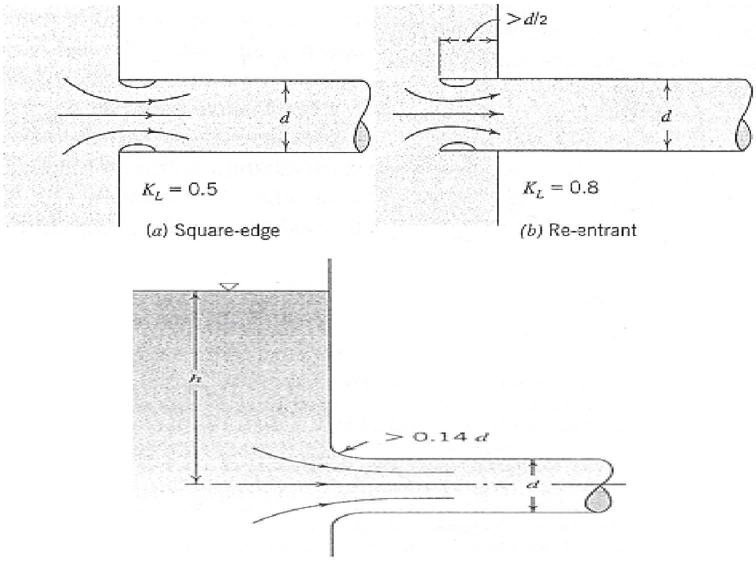
TABLE 2 Contraction, C_c, and Loss, K_L, Coefficients for Abrupt Contractions

A_2/A_1	0	0.1	0.2	0.3	0.4	0.5	0.6	0.7	0.8	0.9	1.0
C_c	0.617	0.624	0.632	0.643	0.659	0.681	0.712	0.755	0.813	0.892	1.00
K_L	0.50	0.46	0.41	0.36	0.30	0.24	0.18	0.12	0.06	0.02	0

The entrance of Fig. 4.11b is known as a re-entrant one. If the pipe wall is very thin and if the plane of the opening is more than one pipe diameter upstream from the reservoir wall, the loss coefficient will be close to 0.8, this high value resulting mainly from the small vena contracta and consequent large deceleration loss. For thick-walled pipes the vena contracta can be expected to be larger and the loss coefficient less than 0.8; Harris17 has shown that pipes of wall thicknesses greater than 0.05J (if square-edged) will give a loss coefficient equal to that of the square-edged entrance.

If the edges of a. pipe entrance are rounded to produce a streamlined bell-mouth (Fig. 4.18) the loss coefficient can be materially reduced. Hamilton18 has shown that any radius of rounding greater than 0.\4d will prevent the formation of a vena contracta and thus eliminate the head loss due to flow deceleration. The nominal value of K_L for such an entrance is about 0.1, but its exact magnitude will depend on the detailed geometry of the entrance and structure of the boundary layer.

The head loss caused by short well-streamlined gradual contractions (Fig. 4.19) is so small that it may usually be neglected in engineering problems. However, an appreciable fall of the hydraulic grade line over such contractions is to be expected; in the pipe downstream from the contraction the hydraulic grade line will be found to slope more steeply



K_L = 0.8 (b) Re-entrant 4.17 Pipe entrances.

than that for established flow because of the change of velocity distribution and boundary layer growth, which cause an increase of a These effects are known to be small and may be ignored in many problems. A nominal value of K_L (for use in Eqn. 4.46 with V — V_2) for short well-streamlined contractions is 0.04; by careful design this figure may be lowered to 0.02, but for long contractions values much larger than 0.04 are to be expected because of extensive wall friction.

Losses of head in smooth pipe bends are caused by the combined effects of separation, wall friction, and the twin-eddy secondary flow described in Section 7.14; for bends of large radius of curvature, the last two effects will predominate, whereas, for small radius of curvature, separation and the secondary flow will be the more significant. The loss of head in a bend is expressed by Eqn. 4.46 in which the head loss is the drop of the extended energy lines (see Fig. 4.13) between the entrance and exit of the bend. Reliable and extensive information on this subject will be found in the work of ltd, 19 a small but typical portion of which is shown in Fig. 4.20. Head loss coefficients for smooth pipe bends provide another example of the

dependence of K_L on shape of passage (determined by \$ and R/d) and Reynolds number; the research of Hofmann20 provides information on the (expected) dependence of loss coefficient on relative roughness as well. To the engineer the most significant feature of head loss in bends is the minimum value of K_L occurring at certain values of R/d which allows selection of bend shapes for maximum efficiency in pipeline design.

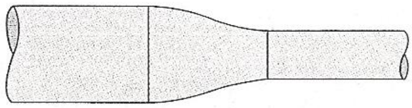


Fig. 9.19 Gradual contraction.

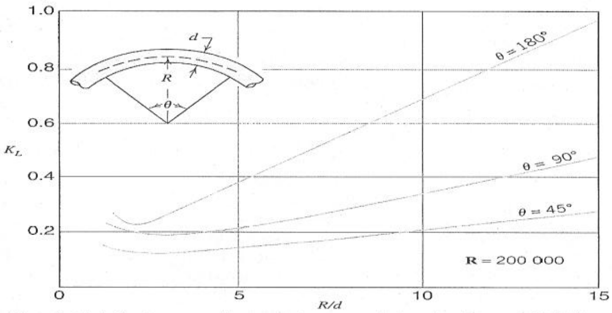


Fig. 9.20 Ito's loss coefficients for smooth bends (R = 200 000).

Tests made on bends with R/d = 0 have given values of K_L around 1.1. Such bends are known as miter bends (Fig. 4.21) and are used widely in large ducts such as wind and water tunnels, where space does not permit a bend of large radius. In these bends, installation of guide vanes materially reduces the head loss and at the same time breaks up the spiral motion and improves the velocity distribution downstream.

The losses of head caused by commercial pipe fittings occur because of their rough and irregular shapes which produce excessive large-scale turbulence. The shapes of commercial pipe fittings are determined more by structural properties ease in handling, and production methods than by head loss considerations, and it is, therefore, not feasible or economically justifiable to build pipe fittings having completely streamlined interiors in order to minimize head loss. The loss of head in commercial pipe fittings is usually expressed by Eqn. 4.46 with V the mean velocity in the pipe and K_L a constant (at high Reynolds numbers), the magnitude of which depends on the shape of the fitting. Values of K_L for various common fittings are available in the Engineering Data Book of the Hydraulic Institute; typical values are presented in Table 3.

It is generally recognized that when fittings are placed in close proximity the total head loss caused by them is less than their numerical sum obtained by the foregoing methods. Systematic tests have not been made on this subject because a simple numerical sum of losses gives a result in excess of the actual losses, and thus, produces an error on the conservative side when design calculations of pressures and flowrate are to be made.

Fig. 9.21 Miter bends.

4.7- PIPELINE PROBLEMS—SINGLE PIPES

All steady-flow pipe problems may be solved by application of the work-energy and continuity equations, and the most effective method of doing this is the construction of energy and hydraulic grade lines. From such lines the variations of pressure, velocity, and unit energy can be clearly seen for the whole problem; thus, the construction of these lines becomes equivalent to writing numerous equations, but the lines lend a clarity to the solution of the problem which equations alone never can.

In engineering offices, tables, charts, monograms, computer software, and so forth, are employed where numerous pipe-flow problems are to be solved. Although all these methods are different, they have their foundations in the work-energy principle, usually with certain approximations; no attempt is made here to cover these many methods—the following discussion will be confined to the application of the work-energy and continuity principles and the use of certain approximations.

Engineering pipe-flow problems usually consist of (1) calculation of head loss and pressure variation from flowrate and pipeline characteristics, (2) calculation of flowrate from pipeline characteristics and the head which produces flow, and (3) calculation of required pipe diameter to pass a given flowrate between two regions of known pressure difference. The first of these problems can be solved directly,

but solution by trial is required for the other two.22 Trial-and-error solutions are necessitated by the fact that the friction factor, /, and loss coefficients, KL, depend on the Reynolds number, which in turn depends on flowrate and pipe diameter, the unknowns of problem types 2 and 3, respectively. However, many engineering pipeline problems involve flow in rough pipes at high Reynolds numbers. Here trial solutions are seldom required (1) because of the tendency of f and K_L toward constancy in this region, (2) because of the inevitable error23 in selecting f from Fig. 9.10, and (3) because engineering answers are usually not needed to a precision which warrants trial-and-error solution in the light of the foregoing facts. Construction of energy and hydraulic grade lines for some typical pipeline problems will indicate further approximations which may frequently be used in the solution of engineering problems.

Consider the calculation of flowrate in a pipeline laid between two tanks or reservoirs having a difference of surface elevation H (Fig. 4.22). The energy line must start in one reservoir surface and end in the other; using a gradual drop to represent head loss due to

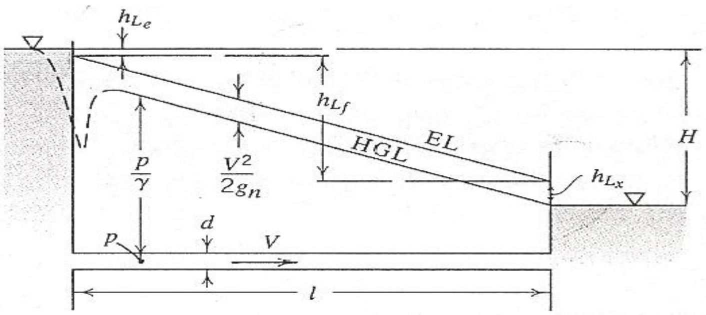


Fig. 9.22

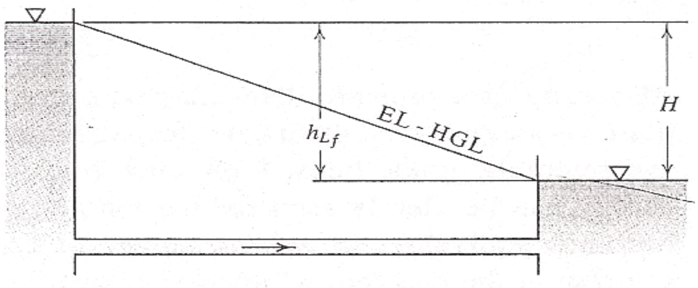


Fig. 9.23

Unless special plots are devised for circumventing this. Due to the inexactness of definition of the roughness.

pipe friction, h_{Lf}, and abrupt drops to represent entrance and exit losses, h_{Lg} and h_L the energy line is constructed as shown. It is apparent from the energy line that

$$h_{L_e} + h_{L_f} + h_{L_x} = H$$

which is the work-energy equation written between the reservoir surfaces. When the appropriate expressions for the head losses are substituted.

$$\left(0.5 + f\frac{l}{d} + 1\right)\frac{V^2}{2g_n} = H$$

If turbulent flow is assumed and a nominal value for f of 0.03 selected (Fig. 4.10), the quantity in parentheses becomes 4.5, 31.5, and 301.5 for-values of 100, 1 000, and 10 000, respectively. The quantities 0.5 and 1.0, which result from inclusion of local losses, have a decreasing effect on the solution with increasing if these terms were omitted entirely, errors of about 18, 2, and 0.3%, respectively, would be produced in the velocity and flowrate. Evidently, the effect of local losses in pipelines of common length is so small that they may often be neglected entirely, appreciably simplifying calculations. Another convenient approximation accompanies the above; increasing l/d also decreases V2/2gn and thus brings the energy line and the hydraulic grade line closer together; since $V^2/2g$ is of the order of the local losses, it is consistent to neglect this also, thus making energy and hydraulic grade lines coincident (except near the entrance) and necessitating the construction of only one line. When the single line is drawn for this pipeline problem (Fig. 4.23), the equation may be written, and the velocity and flowrate may be obtained by trial-and-error procedure.

$$h_L = f\frac{l}{d}\frac{V^2}{2g_n} = H$$

The foregoing approximations are convenient in engineering problems but, of course, cannot be applied blindly and without some experience; preliminary calculations similar to those above will usually indicate the effect of such approximations on the accuracy of the result.

EXAMPLE.15

A clean cast iron pipeline 0.30 m in diameter and 300 m long connects two reservoirs having surface elevations 60 m and 75 m. Calculate the flowrate through this line, assuming water at 10°C and a square-edged entrance.

SOLUTION

Before using the work-energy equation to calculate the flowrate, we need to make some preliminary calculations. The Reynolds number depends on the unknown velocity but we will introduce all the known quantities into the expression for $\mathbf{R}$. From Appendix 2, for water at 20°C, $\nu = 1.306 \times 10^{-6}$ m²/s.

$$\mathbf{R} = \frac{Vd}{\nu} = \frac{V \times 0.30 \text{ m}}{1.306 \times 10^{-6} \text{ m}^2/\text{s}} = 229\,000\ V$$

From Fig. 9.11, $e/d = 0.000\,83$ for clean cast iron pipe. To arrive at a reasonable estimate of the Reynolds number so that a good first estimate of the f-value can be found, we assume $V \approx 2$ m/s. Inserting this number into the expression for $\mathbf{R}$,

$$\mathbf{R} = 229\,000 \times 2 = 458\,000$$

From Fig. 9.10 with $e/d = 0.000\,83$ and $\mathbf{R} = 458\,000$, we find $f = 0.020$. Now, we use this value along with the local loss coefficients in the work-energy equation. Those local loss coefficients are:

$$K_L = 0.5 \text{ for a square-edged entrance}$$

$$K_L = 1.0 \text{ for exit into a reservoir}$$

We are now prepared to utilize the work-energy Eq. 7.35.

$$z_1 + \frac{p_1}{\gamma} + \frac{V_1^2}{2g_n} = z_2 + \frac{p_2}{\gamma} + \frac{V_2^2}{2g_n} + h_L \tag{7.35}$$

Note that the choice of points 1 and 2 at the surface of the two reservoirs means that $p_1 = p_2 = 0$ and $V_1 = V_2 = 0$.

$$75 + 0 + 0 = 60 + 0 + 0 + \left(0.5 + f\frac{l}{d} + 1.0\right)\frac{V^2}{2g_n}$$

$$15 = \left(0.5 + \frac{0.020 \times 300 \text{ m}}{0.30 \text{ m}} + 1.0\right)\frac{V^2}{2 \times 9.81} = 1.096\ V^2$$

$$V = 3.70 \text{ m/s}$$

This is considerably above our estimate so we must re-compute $\mathbf{R}$, find a new f-value and find V again through the work-energy equation. With $V = 3.70$ m/s, $\mathbf{R} = 847\,250$. From Fig. 9.10, $f = 0.019\,3$. Substituting this value into the work-energy equation yields

$$V = 3.76 \text{ m/s}$$

This is clearly close enough to the first calculated value that we can be confident another iteration is unnecessary. The flowrate is

$$Q = VA = 3.76 \text{ m/s} \times (\pi/4)(0.30 \text{ m})^2 = 0.266\text{m}^3/\text{s}$$

EXAMPLE.16

A smooth PVC pipeline 200 ft long is to carry a flowrate of 0.1 ft³/s between two water tanks whose difference in surface elevation is 5 ft. If a square-edged entrance and water at 50°F are assumed, what diameter of pipe is required.

SOLUTION

This type of problem represents the most difficult to solve because the unknown diameter renders both the Reynolds number and the relative roughness as unknown. The only simplification in the case of smooth pipes is that the relative roughness is irrelevant, i.e., we are on the smooth pipe line on the Moody diagram.

We again turn to Appendix 2 to obtain the ν-value of 1.41×10^{-5} ft²/s for water at 50°F. We can also use the technique of representing the Reynolds number in terms of the flowrate as

$$\mathbf{R} = \frac{Vd}{\nu} = \frac{Qd}{A\nu} = \frac{Q}{(\pi/4)\, d \times \nu} = \frac{0.1 \text{ ft}^3/\text{s}}{(\pi/4)\, d \times 1.41 \times 10^{-5} \text{ ft}^2/\text{s}} = \frac{9\,020}{d}$$

Following the previous illustrative problem and applying the work-energy equation, we get

$$5 = \left(0.5 + f\,\frac{200}{d} + 1.0\right)\frac{V^2}{2g_n}$$

We are now involved in a trial-and-error solution for V. The best procedure is to set up a table for the solution process.

Assume d	**R**	f	V	$V^2/2g_n$	(. . . . . .)	**Right-Hand Side**
0.25	36 000	0.022	2.04	0.064 4	19.1	1.23
0.20	45 100	0.021 2	3.18	0.157	22.7	3.56
0.18	50 100	0.020 8	3.93	0.240	24.6	5.90
0.187	48 200	0.021 0	3.64	0.206	24.0	4.94

The value of $d = 0.187$ ft produces a reasonable match to the left-hand side of the equation ($\approx$1% error). Hence, we will have a diameter of $d = 2.24$ inches, which means we will specify the next larger nominal diameter $d = 2.5$ inches. ●

Another interesting application of the work-energy principle occurs when a pipeline extending from a reservoir terminates in a nozzle. This situation for cases where the main line velocity head is both significant and negligible is shown in Fig. 9.24. However, in either case, the velocity head at the nozzle exit cannot be neglected. In the most general case where we will consider local losses, the work-energy equation is

$$z_o + \frac{p_o}{\gamma} + \frac{V_o^2}{2g_n} = z_2 + \frac{p_2}{\gamma} + \frac{V_2^2}{2g_n} + h_L$$

1) *Crude oil at 20 °c flows in riveted steel pipe 1.00 m in diameter at a mean velocity of 2.0 m/s. What range of head loss is to be expected in 1 km of pipelines? For Crude oil at 20°c: $\nu=8.391 * 10^{-6}\ m^2/s$*
2) *Calculate the loss of head and the pressure drop when air at an absolute pressure of 101.3 kPa and 15 °c ($\rho=1.225\ kg/m^3$) flows through 600 m of 450 mm by 300 mm smooth rectangular duct with a mean velocity of 3 m/s.*
   For air at 15°c: $\nu=1.46 * 10^{-5}\ m^2/s$
3) *A clean cast iron pipeline 0.30 m in diameter and 300 m long connects two reservoirs having surface elevation 60 m and 75 m. calculate the flow rate through this line, assuming water at 10 °c and a square – edged entrance.*
   For water at 10°c: $\nu=1.306 * 10^{-6}\ m^2/s$
4) *A smooth PVC pipeline 200 m long is to carry a flow rate 0.05 m^3/s between two tanks whose difference in surface elevation is 8 m. If a square-edged entrance and water at 10 °c are assumed, what diameter of pipe is required? For water at 10 °c : $\nu=1.306 * 10^{-6}\ m^2/s$*
5) *A clean cast iron pipeline 0.26 m in diameter and 200 m long connects two reservoirs having surface elevation 60 m and 160 m. if given: e/d=0.001, $\nu_w=1.3 * 10^{-6}\ m^2/s$, $Re>2 * 10^6$,moody diagram, $K_L=0.5$ for a square edge entrance, $K_L=1$ for exit into reservoir ,*
   - Calculate the flow rate through this pipeline.
   - Check the flow type in the above pipeline.